AutoCAD 2010

User's Guide

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Contents

Get Information

Get Information from Drawings

1

You can retrieve general information from a drawing including identifying information and the number of objects that it contains.

There are types of information stored in a drawing that are not specific to objects within the drawing, but provide useful information to help you understand the behavior of the drawing, the settings of system variables, the number of objects, descriptive information, and so on.

Obtain General Drawing Information

You can retrieve general information about the drawing file and its settings.

This information includes the following:

- Custom descriptive information about the drawing (DWGPROPS)
- General drawing settings (STATUS)
- Amount of time spent in the drawing (TIME)

This information can help you document a drawing, displays a variety of drawing settings such as the total number of objects in the drawing and the amount of free space on your disk drive, and the total amount of time spent in the drawing file.

See also:

- Enter System Variables on the Command Line on page 42
- Add Identifying Information to Drawings on page 188
- Extract Geometric Information from Objects on page 765

- Compare Dimension Styles and Variables on page 1632

Quick Reference

Commands

DWGPROPS

Sets and displays the file properties of the current drawing.

SETVAR

Lists or changes the values of system variables.

STATUS

Displays drawing statistics, modes, and extents.

TIME

Displays the date and time statistics of a drawing.

System Variables

CDATE

Stores the current date and time in decimal format.

DATE

Stores the current date and time in Modified Julian Date format.

SAVENAME

Displays the file name and directory path of the most recently saved drawing.

Utilities

No entries

Command Modifiers

No entries

Count Objects Within a Drawing

You can count objects within a drawing using the QSELECT command.

The QSELECT command displays the Quick Select dialog box, which allows you to create a selection set based on the filtering criteria. You can filter selection sets by property such as color or linetype, and by object type.

Creating a selection set based on the filtering criteria in the Quick Select dialog box, allows you to count specified types of objects within a drawing.

To count specified types of objects in a drawing

1. Click Home tab ➤ Utilities panel ➤ Quick Select. 
2. In the Quick Select dialog box, do one of the following:
   - In the Apply To List, select Entire Drawing.
   - Click the Select Objects button to select a group of objects. Press ENTER. In the Apply To list, select Current selection.
3. In the Object Type list, select the type of object you want to count.
4. In the Properties list, select a property that belongs to the type of objects you want to count.
5. In the Operator list, select = Equals.
6. In the Value list, select the property value of the type of objects you want to count.
7. Click OK.

The number of objects displays at the command prompt.

Menu: Tools ➤ Quick Select
Command entry: QSELECT
Shortcut menu: End any active commands, right-click in the drawing area, and choose Quick Select.

Quick Reference

Commands

QSELECT

Creates a selection set based on filtering criteria.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

The User Interface

Tools in the Application Window

2

Use the Application menu, Quick Access toolbar, and ribbon to access many frequently used commands.

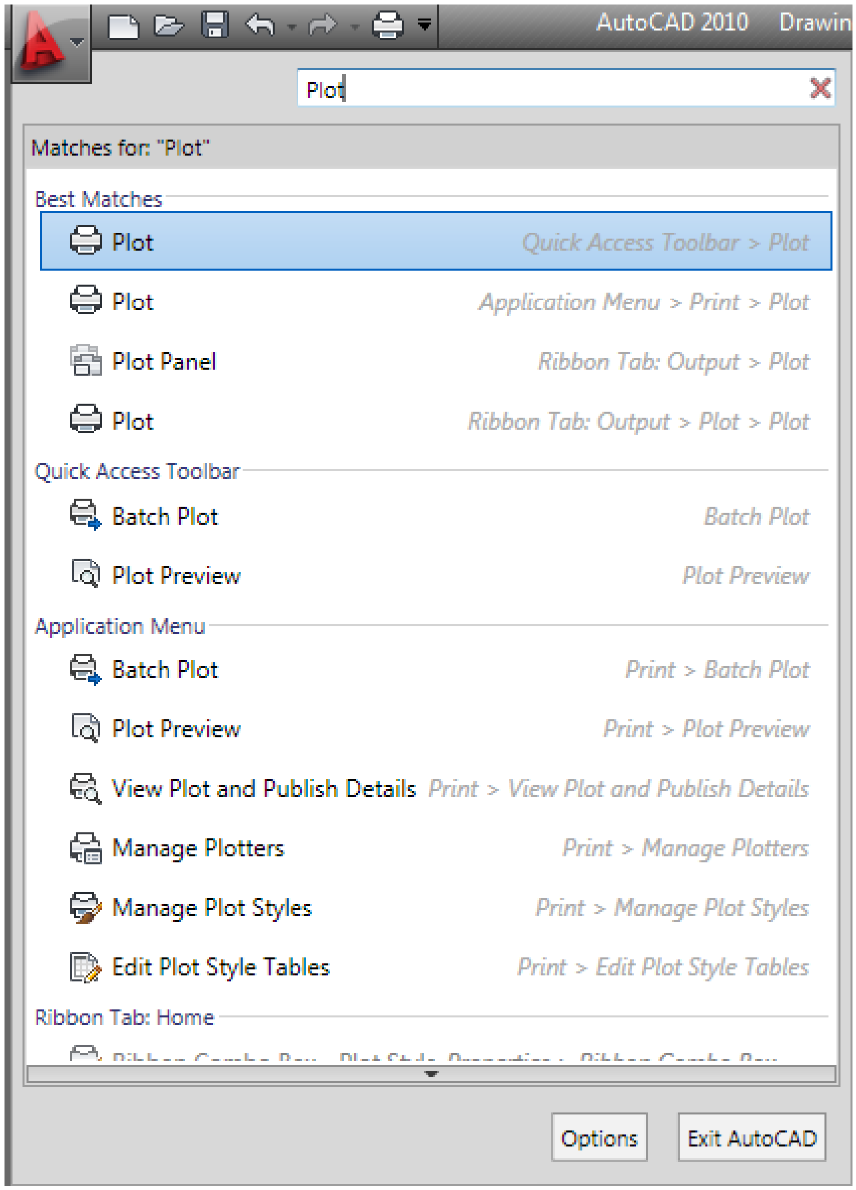
The Application Menu

Click the application button to search for commands, as well as access tools to create, open, and publish a file.

Search for Commands

Perform a real-time search for commands on the Quick Access toolbar, in the application menu, and on the ribbon.

The Search field displays at the top of the application menu. Search results can include menu commands, basic tooltips, and command prompt text strings. You can enter a search term in any language.



To clear the search field in the Application menu

1 Click the Application button.

2 In the Application menu, enter text in the search field.

The spyglass button to the right of the search field becomes an [X].

3 Click the [X] icon to the right of the search field. The search field is cleared and the [X] icon returns to a spyglass.

Quick Reference

Commands

CUI

Manages the customized user interface elements in the product.

OPTIONS

Customizes the program settings.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Access Common Tools

Access common tools to start or publish a file in the application menu.

Click the application button to quickly

- Create, open, or save a file
- Audit, recover, and purge a file
- Print or publish a file
- Access the Options dialog box
- Close AutoCAD

NOTE You can also close AutoCAD by double-clicking the Application button.

Quick Reference

Commands

CUI

Manages the customized user interface elements in the product.

OPTIONS

Customizes the program settings.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

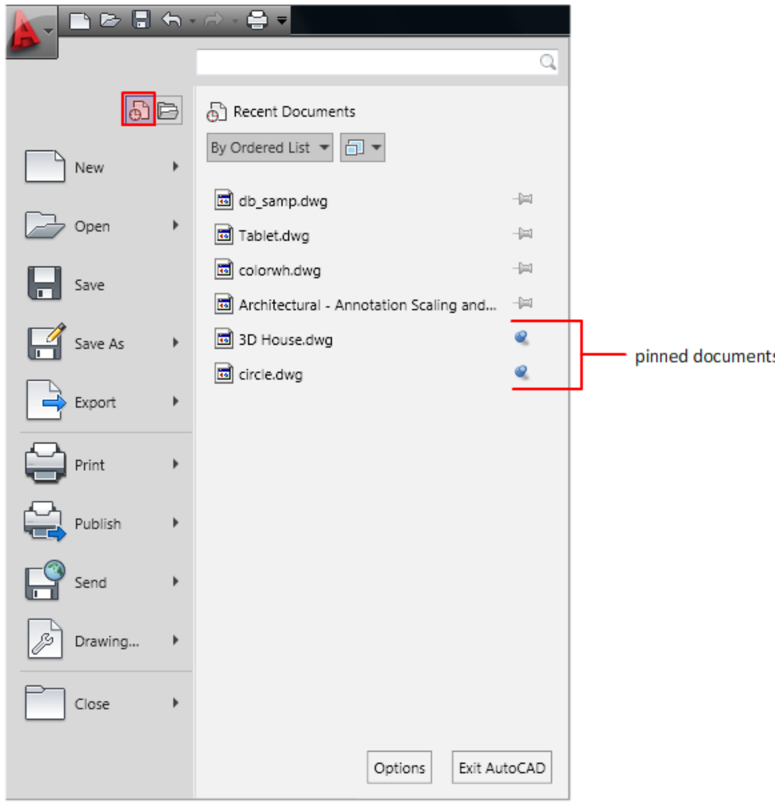
Browse Files

View, sort, and access supported files that you have recently opened.

Recent Documents

View the most recently used files with the Recent Documents list.

Files display in the Recent Documents list with the most recently used file at the top by default.



Pinned Files

You can keep a file listed regardless of files that you save later using the push pin button to the right. The file is displayed at the bottom of the list until you turn off the push pin button.

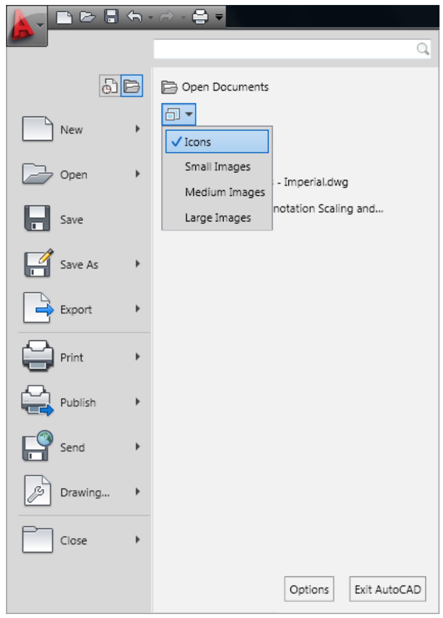
Sort and Group Options

Use the drop-down list at the top of the Recent Documents list to sort or group files by

- File name
- File size
- File type
- Date the files were last modified

To change the preview display options for recent documents

1. Click the Application menu and then, click Recent Documents.
2. Under the Search text box, click the Display Options menu.
3. Select a display option.



NOTE The preview display option you choose remains in both the Recent Documents and Open Documents lists.

To change the number of recent documents listed

1. Click Tools ➤ Options.
2. In the Options dialog box, click the Open and Save tab.
3. In the Application Menu ➤ Number of Recently Used Files text box, enter the number of recent documents to be listed. You can choose any number between 0 and 50.

To keep a document in the Recent Documents list

- Click the push pin button to the right of the document.

To view the Recent Documents list by access date

- In the top-left corner of the Recent Documents list, in the By Ordered List drop-down list, select By Access Date.

To view the Recent Documents list by size

- In the top-left corner of the Recent Documents list, in the By Ordered List drop-down list, select By Size.

To view the Recent Documents list by type

- In the top-left corner of the Recent Documents list, in the By Ordered List drop-down list, select By Type.

Quick Reference

Commands

CUI

Manages the customized user interface elements in the product.

OPTIONS

Customizes the program settings.

System Variables

No entries

Utilities

No entries

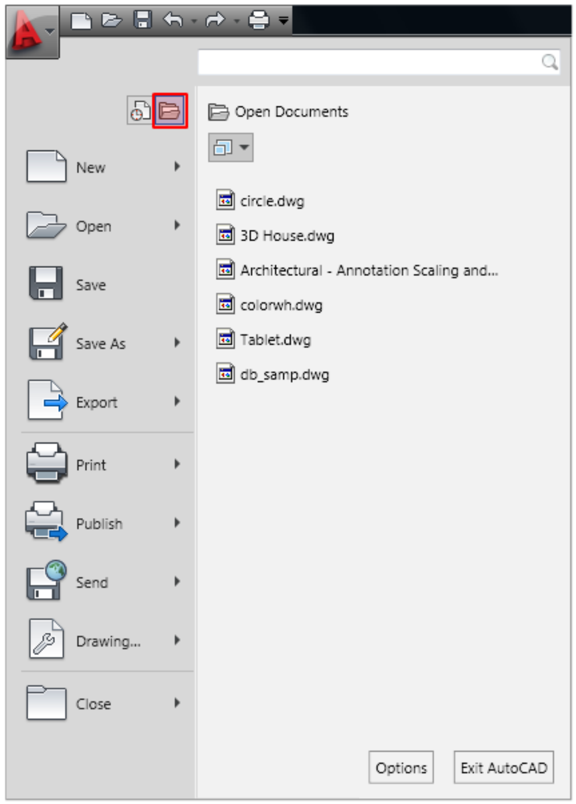
Command Modifiers

No entries

Currently Open Documents

View only files that are currently open with the Open Documents list.

Files display in the Open Documents list with the most recently opened file at the top. To make a file current, click the file in the list.

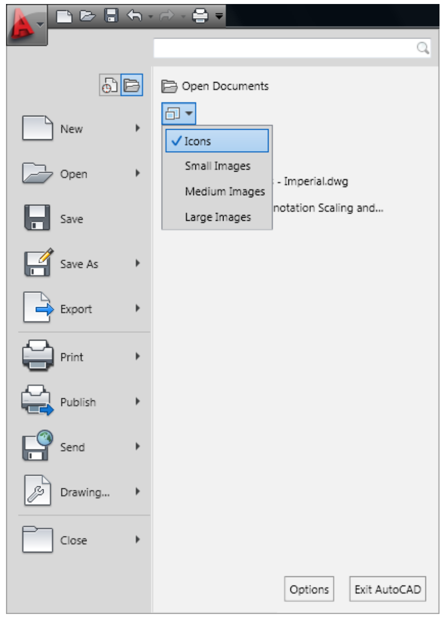


To change the preview display options for currently open documents

1 Click the Application menu and then, click Open Documents.

2 Under the Search text box, click the Display Options menu.

3 Select a display option.



NOTE The preview display option you choose remains in both the Recent Documents and Open Documents quick menus.

Quick Reference

Commands

CUI

Manages the customized user interface elements in the product.

OPTIONS

Customizes the program settings.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Preview Documents

View a thumbnail of files in the Recent Documents and Open Documents lists.

When you hover over a file in either of the lists, a preview of the file is displayed along with the following information:

- Path where the file is stored
- Date the file was last modified
- Version of the product used to create the file
- Name of the person who last saved the file
- Name of the person who is currently editing the file

You can also include a thumbnail of the file next to the files in the list. To change the file icon to a thumbnail preview, click the drop-down list at the top of the Recent Documents or Open Documents lists and choose small icons, large icons, small images, or large images.

Quick Reference

Commands

CUI

Manages the customized user interface elements in the product.

OPTIONS

Customizes the program settings.

System Variables

No entries

Utilities

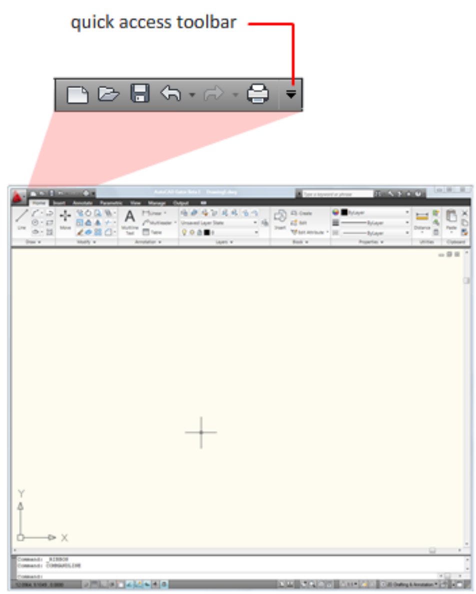
No entries

Command Modifiers

No entries

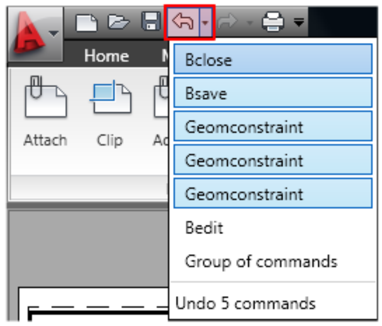
Quick Access Toolbar

Display frequently used tools with the Quick Access toolbar.



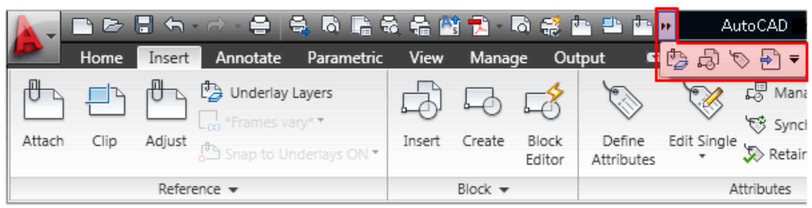
View Undo and Redo History

The Quick Access toolbar displays options to undo and redo changes to your file. To undo or redo a less recent change, click the drop-down button to the right of the Undo and Redo buttons.



Add Commands and Controls

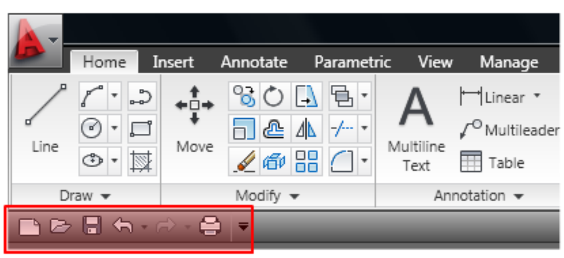
Add unlimited tools to the Quick Access toolbar. Tools that extend past the maximum length of the toolbar are displayed in a flyout button.



To add a ribbon button to the Quick Access toolbar, right-click the button on the ribbon and click Add to Quick Access toolbar. Buttons are added to the right of the default commands on the Quick Access toolbar.

Move the Quick Access Toolbar

Place the Quick Access toolbar either above or below the ribbon using the Customization button.



See also:

- Quick Access Toolbars

- Toolbars on page 32

To add a command to the Quick Access toolbar

1. Right-click the Quick Access toolbar.
2. Click Customize Quick Access Toolbar.
3. In the Customize User Interface (CUI) Editor, drag a command from the Command list to the position you want it to be displayed on the Quick Access toolbar.

   For multiple commands, hold down CTRL and select the commands.
4. Click OK.

   NOTE If you do not click OK, the commands added are not saved to the Quick Access toolbar.

To remove a command from the Quick Access toolbar

1. Right-click over the command you want to remove.
2. Click Remove from Quick Access Toolbar.

Quick Reference

Commands

CUI

Manages the customized user interface elements in the product.

System Variables

No entries

Utilities

No entries

Command Modifiers

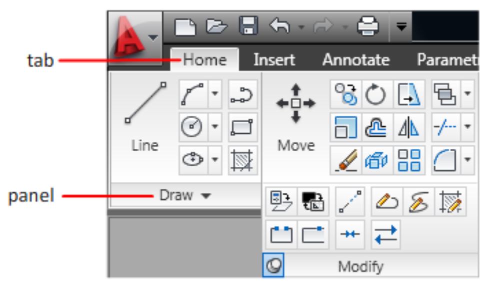
No entries

The Ribbon

The ribbon is a palette that displays task-based commands and controls.

Overview of the Ribbon

The ribbon is displayed automatically when you create or open a file, providing a compact palette of all of the tools necessary to create your file.



The ribbon contains many of the same commands that were once available on the dashboard. For example, the DIMLINEAR command was once available on the dashboard's Dimensions control panel. On the ribbon, DIMLINEAR is located on the Annotation tab, in the Dimensions panel.

Quick Reference

Commands

CUI

Manages the customized user interface elements in the product.

RIBBON

Opens the ribbon window.

RIBBONCLOSE

Closes the ribbon window.

System Variables

MTEXTTOOLBAR

Controls the display of the Text Formatting toolbar.

RIBBONCONTEXTSELECT

Controls how ribbon contextual tabs are displayed when you single- or double-click an object.

RIBBONCONTEXTSELLIM

Suppresses the display of ribbon contextual tabs when the selection set includes more than the specified number of objects.

RIBBONSELECTMODE

Determines whether a pickfirst selection set remains selected after a ribbon contextual tab is invoked and the command is completed.

RIBBONSTATE

Indicates whether the ribbon palette is open or closed.

TABLETOOLBAR

Controls the display of the Table toolbar.

Utilities

No entries

Command Modifiers

No entries

Display and Organize the Ribbon

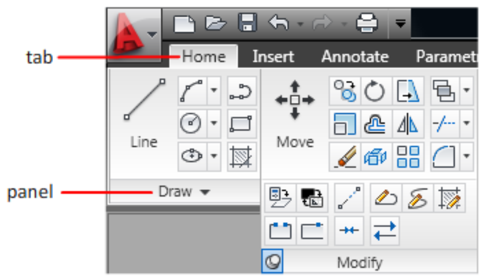
The ribbon is displayed horizontally or vertically.

The horizontal ribbon is displayed across the top of the file window. You can dock the vertical ribbon to the left or right of the application window.

The vertical ribbon can also float in the file window or on a second monitor.

Ribbon Tabs and Panels

The ribbon is composed of a series of panels, which are organized into tabs labeled by task. Ribbon panels contain many of the same tools and controls available in toolbars and dialog boxes.

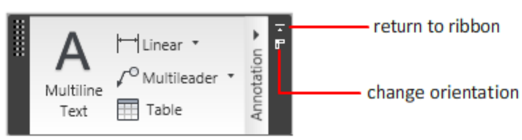


Some ribbon panels display a dialog box related to that panel. An icon in the lower-right corner of the panel indicates that you can display a related dialog box. Click the icon to display the associated dialog box.

To specify which ribbon tabs and panels are displayed, right-click the ribbon and, on the shortcut menu, click or clear the names of tabs or panels.

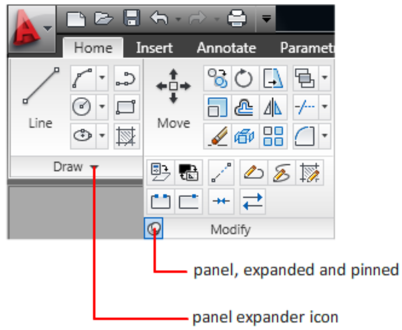
Floating Panels

If you pull a panel off of a ribbon tab and into the drawing area or onto another monitor, that panel floats where you placed it. The floating panel remains open until you return it to the ribbon, even if you switch ribbon tabs.



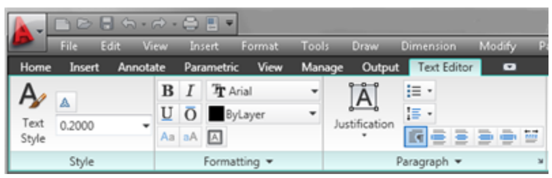
Expanded Panels

An arrow to the right of a panel title indicates that you can expand the panel to display additional tools and controls. By default, an expanded panel closes automatically when you click another panel. To keep a panel expanded, click the push pin icon in the bottom-left corner of the expanded panel.



Contextual Ribbon Tabs

When you execute some commands, a special contextual ribbon tab is displayed instead of a toolbar or dialog box. The contextual tab is closed when you end the command.



See also:

- Create Task-Based Workspaces on page 151

To display the ribbon

- Click Tools menu ➤ Palettes ➤ Ribbon.

  NOTE The ribbon displays the ribbon panels associated with the workspace you used last.

  To display the ribbon panels associated with a specific workspace, click Tools menu ➤ Workspaces.

Command entry: RIBBON

To close the ribbon

- At the Command prompt, enter **ribbonclose**.

Command entry: RIBBONCLOSE
Shortcut menu: Right-click the ribbon tab bar and click Close.

To minimize the ribbon

1. Click the ribbon minimize button to the right of the ribbon tabs.
2. The minimize behavior cycles through the following minimize options:
   - **Minimize to Tabs:** Minimizes the ribbon so that only tab titles are displayed.
   - **Minimize to Panel Titles:** Minimizes the ribbon so that only tab and panel titles are displayed.
   - **Show Full Ribbon:** Displays tabs and full panels, including controls.

Shortcut menu: Right-click the ribbon, click Minimize, and then click one of the above minimize options.
Pointing device: Double-click the name of the active ribbon tab.

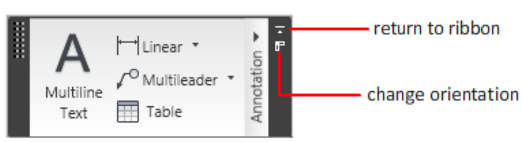
To display or hide a ribbon panel

- Right-click anywhere inside the ribbon. Under Panels, click or clear the name of a panel.

NOTE In the 3D Modeling workspace, both the Home and Render tabs include additional panels that are turned off by default.

To return a floating panel to the ribbon

- Hover over the right side of the floating panel and click the Return Panels to Ribbon icon.



Quick Reference

Commands

CUI

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RIBBON

Opens the ribbon window.

RIBBONCLOSE

Closes the ribbon window.

System Variables

MTEXTTOOLBAR

Controls the display of the Text Formatting toolbar.

RIBBONCONTEXTSELECT

Controls how ribbon contextual tabs are displayed when you single- or double-click an object.

RIBBONCONTEXTSELLIM

Suppresses the display of ribbon contextual tabs when the selection set includes more than the specified number of objects.

RIBBONSELECTMODE

Determines whether a pickfirst selection set remains selected after a ribbon contextual tab is invoked and the command is completed.

RIBBONSTATE

Indicates whether the ribbon palette is open or closed.

TABLETOOLBAR

Controls the display of the Table toolbar.

Utilities

No entries

Command Modifiers

No entries

Customize the Ribbon

You can customize the ribbon depending on your needs.

You can customize the ribbon in the following ways:

- You can create and modify ribbon panels using the Customize User Interface Editor. See Ribbon in the *Customization Guide*.
- You can associate a customizable tool palette group with each tab on the ribbon. Right-click the ribbon tab to display a list of available tool palette groups.
- You can change the order of ribbon tabs. Click the tab you want to move, drag it to the desired position, and release.
- You can change the order of ribbon panels. Click the panel you want to move, drag it to the desired position, and release.
- You can convert toolbars into ribbon panels using the Customize User Interface Editor. See Ribbon in the *Customization Guide*.

See also:

- Ribbon

To associate a tool palette group with a ribbon tab

1. Click Manage Tab ➤ Customization panel ➤ User Interface. 

2. In the Customize User Interface (CUI) Editor, Customize tab, in the Customizations In <*file name*> pane, click the plus sign (+) next to the Workspaces node to expand it.
3. Select the workspace that has the ribbon tab for which you want to assign a tool palette group.

4 In the Workspace Contents pane, click the plus sign (+) next to the Ribbon Tabs node to expand it.

5 Select the ribbon tab that you want to assign a tool palette group.

6 In the Properties pane, in the ToolPalette Group box, click the down arrow and select the tool palette group you want to assign to the ribbon tab.

7 Click OK.

To display the tool palette group associated with a ribbon tab

- Right-click a ribbon tab and click Show Related Tool Palette Group.

Quick Reference

Commands

CUI

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RIBBONSTATE

Indicates whether the ribbon palette is open or closed.

TABLETOOLBAR

Controls the display of the Table toolbar.

Utilities

No entries

Command Modifiers

No entries

Other Tool Locations

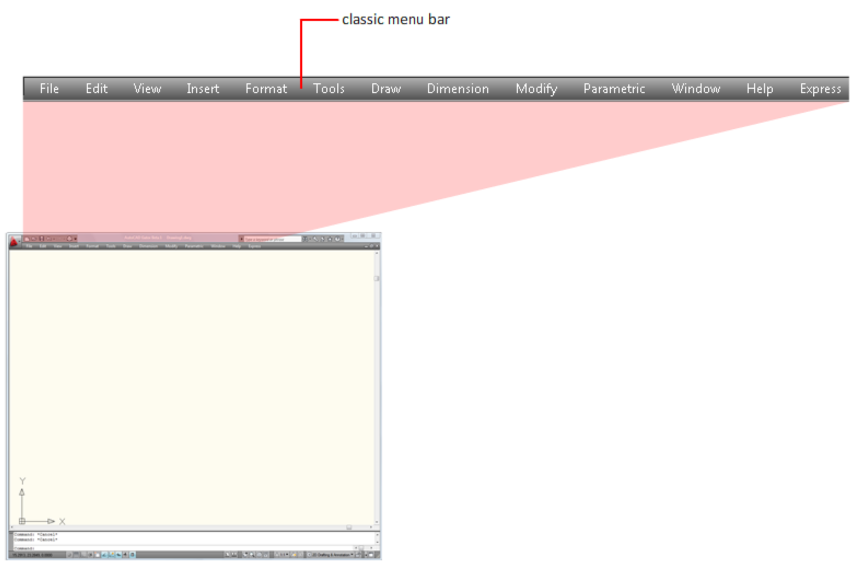
3

Use common tools in the classic menu bar, toolbars, tool palettes, status bars, shortcut menus, and Design Center to find more commands, settings, and modes.

Access the Classic Menu Bar

Display pull-down menus from the classic menu bar using one of several methods. You can also specify alternate menus.

The classic menu bar can be displayed at the top of the drawing area. To display the classic menu bar,



You can specify menus to display in the menu browser for all workspaces by customizing a CUIx file and loading it into the program.

See also:

- Create Task-Based Workspaces on page 151
- "Pull-down and Shortcut Menus" in the *Customization Guide*

To display the classic menu bar

- On the Quick Access toolbar, click the Customization drop-down menu ➤ Show Menu Bar.

Command entry: MENUBAR

Quick Reference

Commands

No entries

System Variables

MENUBAR

Controls the display of the menu bar.

Utilities

No entries

Command Modifiers

No entries

Toolbars

Use buttons on toolbars to start commands, display flyout toolbars, and display tooltips. You can display or hide, dock, and resize toolbars.

Toolbars contain buttons that start commands. When you move your mouse or pointing device over a toolbar button, the tooltip displays the name of the button. Buttons with a small black triangle in the lower-right corner are flyout toolbars that contain related commands. With the cursor over the icon, hold down the left button on your mouse until the flyout toolbar displays.

The Quick Access toolbar at the top of the application window is displayed by default. This toolbar is like those found in Microsoft® Office programs. It contains frequently used AutoCAD® commands such as PLOT, UNDO, and REDO, as well as Microsoft Office standard commands such as New, Open, and Save. For more information about the Quick Access toolbar, see Quick Access Toolbar on page 19.

NOTE You can turn a toolbar into a ribbon panel using the Customize User Interface dialog box.

Display or Hide, Dock, and Resize Toolbars

You can display or hide toolbars, and you can save your selections as a workspace. You can also create your own toolbars.

A toolbar displays as *floating* or *docked*. A floating toolbar displays anywhere in the drawing area, and you can drag a floating toolbar to a new location, resize it, or dock it. A docked toolbar is attached to any edge of the drawing area. A toolbar docked at the top edge of the drawing area is located below the ribbon. You can move a docked toolbar by dragging it to a new docking location.

See also:

- Create Task-Based Workspaces on page 151
- “Toolbars” in the *Customization Guide*
- Ribbon in the *Customization Guide*

To display a toolbar

Display a toolbar with one of the following methods

- Right-click any toolbar and click a toolbar on the shortcut menu.
- Click Tools ➤ Toolbars and click the toolbar to display.

To dock a toolbar

1. Position the cursor on the name of the toolbar or in any blank area, and hold down the button on your pointing device.
2. Drag the toolbar to a docking location at the top, bottom, or either side of the drawing area.

3 When the outline of the toolbar is displayed in the docking area, release the button.

   To place a toolbar in a docking region without docking it, hold down Ctrl as you drag.

To undock a toolbar

1 Position the cursor on the double bars at the end of the toolbar, and hold down the button on your pointing device.

2 Drag the toolbar away from its docked location and release the button.

To resize a toolbar

1 Position the cursor on the edge of a floating toolbar until the cursor changes to a horizontal or vertical double arrow.

2 Hold down the button and move the cursor until the toolbar is in the shape you want.

To close a toolbar

1 If the toolbar is docked, undock it.

2 Click the Close button in the upper-right corner of the toolbar.

Quick Reference

Commands

CUI

Manages the customized user interface elements in the product.

System Variables

TOOLTIPS

Controls the display of tooltips on the ribbon, toolbars, and other user interface elements.

Utilities

No entries

Command Modifiers

No entries

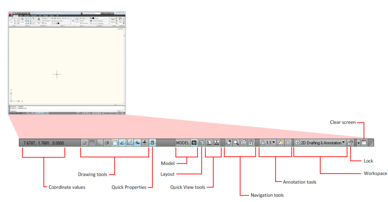
Status Bars

The application and drawing status bars provide useful information and buttons for turning drawing tools on and off.

Application Status Bar

The application status bar displays the coordinate values of your cursor, drawing tools, navigation tools, and tools for Quick View and annotation scaling.

You can view the drawing tool buttons as icons or text. You also can easily change the settings of snap, polar, osnap, and otrack from the shortcut menus of these drawing tools.



You can preview and switch between open drawings and layouts in a drawing. Use the navigation tools to switch between open drawings and view a model in a drawing. You can also display tools for scaling annotations.

With the Workspace button, you can switch workspaces. The lock button locks the current positions of the toolbars and windows. To expand the drawing display area, click the Clean Screen button.

You can add or remove a button from the application status bar from the shortcut menu of the status bar.

NOTE When the application status bar is turned off, the Clean Screen button is not displayed on the screen.

To display the application status bar

- At the Command prompt, enter STATUSBAR and enter **1** to display only the application status bar, or **2** to display both the application and drawing status bars.

To control the display of icons and notifications in the status bar tray

1. Click View tab ➤ Windows panel ➤ Status Bar.
2. In the Status Bar drop-down, click Tray Settings.
3. In the Tray Settings dialog box, select or clear the following display options:
   - **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 such as Communications Center. When the Display Icons from Services option is cleared, this option is unavailable.
4. If Display Notifications from Services is selected, set a time for a notification to be displayed, or select Display Until Closed.
5. Click OK.

Command entry: TRAYSETTINGS

To control the display of buttons on the status bar

1. Click View tab ➤ Windows panel ➤ Status Bar.
2. In the Status Bar drop-down, select any button name to change the display.

Command entry: TRAYSETTINGS

To control the display of cursor coordinates on the status bar

1 Click View tab ➤ Windows panel ➤ Status Bar.
2 In the Status Bar drop-down, select or clear Cursor Coordinate Values.

Command entry: TRAYSETTINGS

Quick Reference

Commands

TRAYSETTINGS

Controls the display of icons and notifications in the status bar tray.

System Variables

STATUSBAR

Controls the display of the application and drawing status bars.

Utilities

No entries

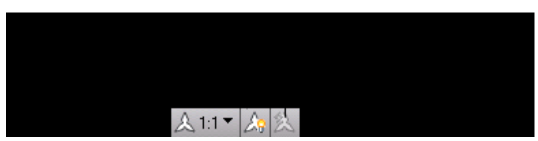
Command Modifiers

No entries

Drawing Status Bar

The drawing status bar displays several tools for scaling annotations.

Different tools display for model space and paper space.



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.

When the drawing status bar is turned on, you can use the Drawing Status Bar menu to select which tool to display on the status bar.

To turn the drawing status bar on or off

- Click View tab ➤ Windows panel ➤ Drawing Status Bar. 

Command entry: STATUSBAR

To control the display of buttons on the drawing status bar

- Click the arrow to the right of the drawing status bar and select or clear any option name.
  Checked items are displayed on the drawing status bar.

Quick Reference

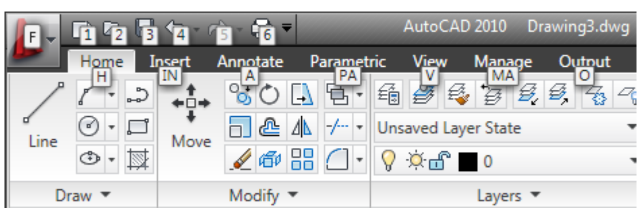
System Variables

STATUSBAR

Controls the display of the application and drawing status bars.

Keytips

Use the keyboard to access the Application menu, Quick Access toolbar, and ribbon.



Press the Alt key to display shortcut keys for common tools in the application window.

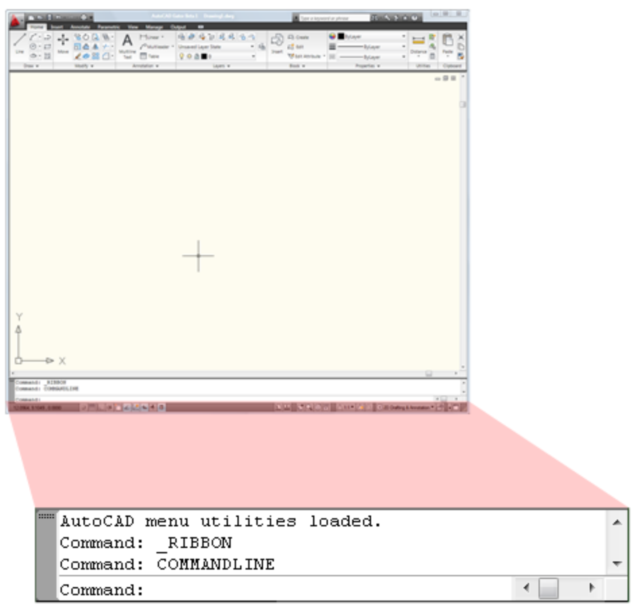
When you select a keytip, more keytips are displayed for that tool.

The Command Window

Enter Commands on the Command Line

You can enter a command by using the keyboard. Some commands also have abbreviated names called *command aliases*.

Commands, system variables, options, messages, and prompts are displayed in a dockable and resizable window called the *command window*. The bottom line of the command window is called the *command line*. The command line displays the operation in progress and provides an inside view of exactly what the program is doing.



To enter a command by using the keyboard, type the full command name on the command line and press Enter or Spacebar.

NOTE When Dynamic Input is on and is set to display dynamic prompts, you can enter many commands in tooltips near the cursor.

Some commands also have abbreviated names. For example, instead of entering **line** to start the LINE command, you can enter **l**. Abbreviated command names are called *command aliases* and are defined in the *acad.pgp* file.

To define your own command aliases, see Create Command Aliases in the *Customization Guide*.

To find a command, you can type a letter on the command line and press TAB to cycle through all the commands that begin with that letter. Press Enter or Spacebar. Restart a recently used command by right-clicking on the command line.

Specify Command Options

When you enter commands on the command line, you see either a set of options or a dialog box. For example, when you enter **circle** at the Command prompt, the following prompt is displayed:

Specify center point for circle or [3P/2P/Ttr (tan, tan, radius)]:

You can specify the center point either by entering *X,Y* coordinate values or by using the pointing device to click a point on the screen.

To choose a different option, enter the letters capitalized in one of the options in the brackets. You can enter uppercase or lowercase letters. For example, to choose the three-point option (3P), enter **3p**.

Execute Commands

To execute commands, press Spacebar or Enter, or right-click your pointing device after entering command names or responses to prompts. The instructions in Help assume this step and do not specifically instruct you to press Enter after each entry.

Repeat and Cancel Commands

If you want to repeat a command that you have just used, press Enter or Spacebar, or right-click your pointing device at the Command prompt.

You also can repeat a command by entering **multiple**, a space, and the command name, as shown in the following example:

Command: **multiple circle**

To cancel a command in progress, press ESC.

Interrupt a Command with Another Command or System Variable

Many commands can be used transparently: that is, they can be entered on the command line while you use another command. Transparent commands frequently change drawing settings or display , for example, GRID or ZOOM.

In the *Command Reference*, transparent commands are designated by an apostrophe in front of the command name.

To use a command transparently, click its toolbar button or enter an apostrophe (') before entering the command at any prompt. On the command line, double angle brackets (>>) precede prompts that are displayed for transparent commands. After you complete the transparent command, the original command resumes. In the following example, you turn on the dot grid and set it to one-unit intervals while you draw a line, and then you continue drawing the line.

Command: **line**
Specify first point: **'grid**
>>Specify grid spacing (X) or [ON/OFF/Snap/Aspect] <0.000>: **1**
Resuming LINE command
Specify first point:

Commands that do *not* select objects, create new objects, or end the drawing session usually can be used transparently. Changes made in dialog boxes that you have opened transparently cannot take effect until the interrupted command has been executed. Similarly, if you reset a system variable transparently, the new value cannot take effect until you start the next command.

See also:

- "Keyboard Shortcuts" in the *Customization Guide*

To copy a command you have recently used

1. Right-click on the command line. Click Recent Commands.
2. Click the command you want to use.

Quick Reference

Commands

MULTIPLE

Repeats the next command until canceled.

OPTIONS

Customizes the program settings.

PASTECLIP

Pastes objects from the Clipboard into the current drawing.

System Variables

CMDNAMES

Displays the names of the active and transparent commands.

Utilities

No entries

Command Modifiers

No entries

Enter System Variables on the Command Line

System variables are settings that control how certain commands work.

They can turn on or turn off modes such as Snap, Grid, or Ortho. They can set default scales for hatch patterns. They can store information about the current drawing and about program configuration. Sometimes you use a system variable in order to change a setting. At other times you use a system variable to display the current status.

For example, the GRIDMODE system variable turns the dot grid display on and off when you change the value. In this case, the GRIDMODE system variable is functionally equivalent to the GRID command. DATE is a read-only system variable that stores the current date. You can display this value, but you cannot change it.

Bitcode Variables

Some system variables are controlled using *bitcodes*. With these system variables, you add values to specify a unique combination of behaviors. For example, the LOCKUI system variable provides the following bitcode 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

Thus, if LOCKUI is set to 1 + 4 = 5, only docked and floating toolbars are locked; docked, anchored, and floating windows are not locked.

NOTE You can examine or change a system variable's setting transparently, that is, while using another command; however, new values may not take effect until the interrupted command ends.

To change the setting of a system variable

1 At the Command prompt, enter the system variable name. For example, enter **gridmode** to change the grid setting.

2 To change the status of GRIDMODE, enter **1** for on or **0** for off. To retain the current value of the system variable, press Enter.

To see a complete list of system variables

1 Click Tools menu ➤ Inquiry ➤ Set Variable..

2 At the Variable Name prompt, enter **?**.

3 At the Enter Variable(s) to List prompt, press Enter.

Command entry: SETVAR

Quick Reference

Commands

SETVAR

Lists or changes the values of system variables.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Navigate and Edit Within the Command Window

You can edit text in the command window to correct or repeat commands.

Use the standard keys:

- Up, Down, Left Arrow, and Right Arrow
- Insert, Delete
- Page Up, Page Down
- Home, End
- Backspace

You can repeat any command used in the current session by cycling through the commands in the command window with Up ArrowCtrl and Down Arrow and pressing Enter. By default, pressing Ctrl+C copies highlighted text to the Clipboard. Pressing Ctrl+V pastes text from the Clipboard to the text window or the command window.

If you right-click in the command window or text window, a shortcut menu is displayed from which you can access the six most recently used commands, copy selected text or the entire command history, paste text, and access the Options dialog box.

For most commands, a command line with two or three lines of previous prompts, called the command history, is sufficient for viewing and editing. To see more than one line of command history, you can scroll through the history or resize the command window by dragging its border. For commands with text output, such as LIST, you might need a larger command window, or you can press F2 to use the text window.

Use the Text Window

The text window is a window similar to the command window in which you can enter commands and view prompts and messages. The text window displays a complete command history for the current work session. Use the text window to view lengthy output of commands such as LIST, which displays detailed information about objects you select. To move forward and backward in the command history, you can click the scroll arrows along the right edge of the window.

Press SHIFT with a key to highlight text. For example, press SHIFT+HOME in the text window to highlight all text from the cursor location to the beginning of the line.

To copy all the text in the text window to the Clipboard, use the COPYHIST command.

To save commands to a log file, use the LOGFILEON command.

See also:

- Dock, Resize, and Hide the Command Window on page 49
- Use Dynamic Input on page 668

To close the text window

- At the Command prompt, enter **graphscr**.

NOTE You can also close the text window by pressing F2, or using the standard Windows controls.

To copy text from the text window to the command line

1. If the text window is not displayed, click Tools tab ➤ Windows Elements panel ➤ Text Window. 

2. Select the text you want to copy.
3. Right-click in the command window or text window. Click Paste to Command Line.

   The text is copied to the Clipboard and then pasted on the command line. After you press Enter, the commands are executed in sequence, like a script. You can also use Ctrl+C and Ctrl+V to copy and paste text.

Command entry: COPYCLIP, PASTECLIP

To display the text window

- Click View tab ➤ Windows Elements panel ➤ Text Window. 

The text window is displayed in front of the drawing area.

Command entry: TEXTSCR

Quick Reference

Commands

COPYCLIP

Copies selected objects to the Clipboard.

COPYHIST

Copies the text in the command line history to the Clipboard.

GRAPHSCR

Switches from the text window to the drawing area.

LOGFILEOFF

Closes the text window log file opened by LOGFILEON.

LOGFILEON

Writes the text window contents to a file.

PASTECLIP

Pastes objects from the Clipboard into the current drawing.

TEXTSCR

Opens the text window.

System Variables

LOGFILEMODE

Specifies whether the contents of the text window are written to a log file.

LOGFILENAME

Specifies the path and name of the text window log file for the current drawing.

LOGFILEPATH

Specifies the path for the text window log files for all drawings in a session.

Utilities

No entries

Command Modifiers

No entries

Switch Between Dialog Boxes and the Command Line

You can display prompts on the command line instead of using a dialog box, or switch back again. This option is useful primarily when using scripts.

Some functions are available both on the command line and in a dialog box. In many cases, you can enter a hyphen before the command to suppress the dialog box and display prompts on the command line instead. For example, entering **layer** on the command line displays the Layer Properties Manager. Entering **-layer** on the command line displays the equivalent command line options. Suppressing the dialog box is useful for compatibility with earlier versions of AutoCAD® and for using script files. There may be slight differences between the options in the dialog box and those available on the command line.

These system variables also affect the display of dialog boxes:

- ATTDIA controls whether INSERT uses a dialog box for attribute value entry.
- CMDNAMES displays the name (in English) of the currently active command and transparent command.
- EXPERT controls whether certain warning dialog boxes are displayed.
- FILEDIA controls the display of dialog boxes used with commands that read and write files. For example, if FILEDIA is set to 1, SAVEAS displays the Save Drawing As dialog box. If FILEDIA is set to 0, SAVEAS displays prompts on the command line. The procedures in this documentation

assume that FILEDIA is set to 1. Even when FILEDIA is set to 0, you can display a file dialog box by entering a tilde (~) at the first prompt.

FILEDIA and EXPERT are useful when you use scripts to run commands.

To use the command line version of a command

- For most commands, enter minus (-) in front of the command.
- For dialog boxes that open and save files, set the FILEDIA system variable to 0.

Quick Reference

Commands

GRAPHSCR

Switches from the text window to the drawing area.

System Variables

ATTDIA

Controls whether the INSERT command uses a dialog box for attribute value entry.

CMDNAMES

Displays the names of the active and transparent commands.

EXPERT

Controls whether certain prompts are issued.

FILEDIA

Suppresses display of file navigation dialog boxes.

Utilities

No entries

Command Modifiers

No entries

Dock, Resize, and Hide the Command Window

Change the position and display of the command window to suit the way you work.

Dock the Command Window

By default, the command window is docked. The docked command window is the same width as the AutoCAD window. If text that is entered becomes longer than the width of the command line, the window pops up in front of the command line to show the full text of the line.

Undock, or float, the command window by dragging it away from the docking region. The docking region is an edge of the AutoCAD application window that allows you to dock a toolbar, palette, or the command window. You can move the floating command window anywhere on the screen and resize its width and height with the pointing device.

Dock a floating command window again by dragging it to the docking region of the AutoCAD window.

Anchoring the Command Window

The command window can be anchored on the left or right side of the AutoCAD window. By anchoring the command window it remains on screen, but in a minimized state, which allows you to bring it back up when needed. This also helps to increase the amount of visible drawing area. The command window must be floating before an anchoring side can be selected. To anchor the command window make sure it is floating, and then right-click over its title bar and select either Anchor Left or Anchor Right.

Resize the Command Window

You can resize the command window vertically by dragging the splitter bar, which is located on the top edge of the window when it is docked on the bottom and at the bottom edge of the window when it is docked at the top.

Hide the Command Window

Hide and redisplay the command line by doing one of the following:

- Click Tools menu ➤ Command Line.
- Press Ctrl+9.

When you hide the command line, you can still enter commands. However, some commands and system variables return values at the command line, so you may want to redisplay the command line in those instances.

NOTE For information about display options (such as auto-hide or transparency) for dockable windows, see "Control the Display of Dockable Windows" in the topic Set Interface Options on page 111.

To float the command window

- Click the move handle (the double bars) on the left edge of the docked command window and drag the command window away from the docking region until it has a thick outline. Then drop it in the drawing area of the AutoCAD window.

To make the floating command window transparent

1. In the floating Command window, click the Properties button and select Transparency.
2. In the Transparency dialog box, move the slider to the left to make the command window less transparent and to the right to make it more transparent.

   The range is from opaque to transparent. When the Turn Off Transparency for All Palettes option is selected, the command window cannot be made transparent.

To dock the command window

- Click the title bar and drag the command window until it is over the top or bottom docking region of the AutoCAD window. When the command window becomes the same width as the AutoCAD window, release the mouse button to dock it.
  The docking region is an edge of the AutoCAD application window that allows you to dock a toolbar, palette, or the command window.

■ Right-click the title bar of the Command window, and select Allow Docking.

NOTE You can control whether the command window docks by right-clicking the title bar of the command window and then clicking Allow docking.

To resize the command window when it is docked

1. Position the cursor over the horizontal splitter bar so that the cursor appears as a double line and arrows.
2. Drag the splitter bar vertically until the command window is the size you want it to be.

To hide the command window

■ Click View tab ➤ Palettes panel ➤ Command Line. 

NOTE Some commands and system variables return values at the command line, so you may want to display the command line in those instances. To display the command line when it's hidden, press Ctrl+9. Alternate method:

Command entry: COMMANDLINE

Quick Reference

Commands

COMMANDLINE

Displays the Command Line window.

COMMANDLINEHIDE

Hides the Command Line window.

System Variables

PALETTEOPAQUE

Controls whether palettes can be made transparent.

Utilities

No entries

Command Modifiers

No entries

Shortcut Menus

Display a shortcut menu for quick access to commands that are relevant to your current activity.

You can display different shortcut menus when you right-click different areas of the screen. Shortcut menus typically include options to

- Repeat the last command entered
- Cancel the current command
- Display a list of recent user input
- Cut, copy, and paste from the Clipboard
- Select a different command option
- Display a dialog box, such as Options or Customize
- Undo the last command entered

You can customize right-click behavior to be time-sensitive, so that a quick right-click acts the same as pressing Enter, and a longer right-click displays a shortcut menu.

Shortcut menus can be customized using a customization (CUIx) file. The main CUIx file is called *acad.cuix* by default.

See also:

- "Create Submenus" in the *Customization Guide*

To display a shortcut menu

1. Move the cursor over an area, feature, or icon.
2. Right-click your mouse, or press the equivalent button on your pointing device.

A shortcut menu relevant to the cursor location is displayed. If one or more objects are selected when you right-click in the drawing area, an editing-oriented shortcut menu is displayed. You can also display a shortcut menu during PAN or ZOOM.

To turn off shortcut menus in the drawing area

1. Click Tools menu ➤ Options.
2. In the Options dialog box, User Preferences tab, under Windows Standard Behavior, clear Shortcut Menus in Drawing Area.
3. To control Default, Edit, and Command shortcut menus individually, select Shortcut Menus in Drawing Area. Right-Click Customization.
4. In the Right-Click Customization dialog box, under Default Mode or Edit Mode, select one of the following options to control what happens when you right-click in the drawing area and no command is in progress:
   - **Repeat Last Command.** Repeats the last command. Selecting this option turns off the Default and Edit shortcut menus. Right-clicking is the same as pressing Enter.
   - **Shortcut Menu.** Displays the Default or Edit shortcut menu.
5. Under Command Mode, select one of the following options to determine what happens when you right-click in the drawing area while a command is in progress:
   - **Enter.** Turns off the Command shortcut menu. Right-clicking is the same as pressing Enter.
   - **Shortcut Menu: Always Enabled.** Displays the Command shortcut menu.
   - **Shortcut Menu: Enabled When Command Options Are Present.** Displays the Command shortcut menu only when options are currently available in the Command prompt. In a Command prompt, options are enclosed in square brackets. If no options are available, right-clicking is the same as pressing Enter.

   In addition to turning the Default, Edit, and Command shortcut menus on and off, you can customize the options that are displayed on them. For example, you can add options to the Edit shortcut menu that are displayed only when circles are selected.

To turn on time-sensitive right-click behavior

1 Click Tools menu ➤ Options.

2 In the Options dialog box, User Preferences tab, under Windows Standard Behavior, click Right-Click Customization.

3 In the Right-Click Customization dialog box, select Turn on Time-Sensitive Right-Click.

   You can specify the duration of the longer click. The default is 250 milliseconds

4 Click Apply & Close.

5 In the Options dialog box, click OK.

Command entry: OPTIONS

To control the display of recent input

1 At the Command prompt, enter **inputhistorymode**.

2 Enter a sum of one or more of the following values:

   - **0**. No history of recent input is displayed.
   - **1**. History of recent input is displayed at the Command prompt with access through Up Arrow and Down Arrow keys.
   - **2**. History of recent input for the current command is displayed in the shortcut menu.
   - **4**. History of recent input for all commands in the current session is displayed in the shortcut menu.
   - **8**. Markers for recent input of point locations are displayed in the drawing.

   The default value is 15.

3 (Optional) At the Command prompt, enter **cmdinputhistorymax**.

4 Enter a value to control how many unique values entered at a prompt are remembered and available to be displayed as recent input.

Quick Reference

Commands

COPYCLIP

Copies selected objects to the Clipboard.

COPYHIST

Copies the text in the command line history to the Clipboard.

CUI

Manages the customized user interface elements in the product.

CUTCLIP

Copies selected objects to the Clipboard and removes them from the drawing.

OPTIONS

Customizes the program settings.

PAN

Moves the view in the current viewport.

PASTECLIP

Pastes objects from the Clipboard into the current drawing.

PROPERTIES

Controls properties of existing objects.

TRAYSETTINGS

Controls the display of icons and notifications in the status bar tray.

U

Reverses the most recent operation.

ZOOM

Increases or decreases the magnification of the view in the current viewport.

System Variables

CMDINPUTHISTORYMAX

Sets the maximum number of previous input values that are stored for a prompt in a command.

INPUTHISTORYMODE

Controls the content and location of the display of a history of user input.

PICKFIRST

Controls whether you select objects before (noun-verb selection) or after you issue a command.

SHORTCUTMENU

Controls whether Default, Edit, and Command mode shortcut menus are available in the drawing area.

TRAYICONS

Controls whether a tray is displayed on the status bar.

TRAYNOTIFY

Controls whether service notifications are displayed in the status bar tray.

TRAYTIMEOUT

Controls the length of time (in seconds) that service notifications are displayed.

Utilities

No entries

Command Modifiers

No entries

Tool Palettes

Tool palettes are tabbed areas within the Tool Palettes window that provide an efficient method for organizing, sharing, and placing blocks, hatches, and other tools. Tool palettes can also contain custom tools provided by third-party developers.

Create and Use Tools from Objects and Images

You can create a tool by dragging objects from your drawing onto a tool palette. You can then use the new tool to create objects with the same properties as the object you dragged onto the tool palette.

Tool palettes are tabbed areas within the Tool Palettes window. The items you add to a tool palette are called *tools*. You can create a tool by dragging any of the following, one at a time, onto your tool palette:

- Geometric objects such as lines, circles, and polylines
- Dimensions
- Blocks
- Hatches
- Solid fills
- Gradient fills
- Raster images
- External references (xrefs)

NOTE When you drag an object onto a tool palette, you can switch to a different tab by hovering over the tab for a few seconds.

You can then use the new tool to create objects in your drawing with the same properties as the object you dragged to the tool palette. For example, if you drag a red circle with a lineweight of .05 mm from your drawing to your tool palette, the new tool creates a red circle with a lineweight of .05 mm. If you drag a block or xref to a tool palette, the new tool inserts the block or xref with the same properties into your drawing.

When you drag a geometric object or a dimension onto a tool palette, the new tool is automatically created with an appropriate flyout. Dimension tool flyouts, for example, provide an assortment of dimension styles. Click the arrow on the right side of the tool icon on the tool palette to display the flyout. When you use a tool on a flyout, the object in the drawing has the same properties as the original tool on the tool palette.

Insert Blocks and Attach References

You can choose to be prompted for a rotation angle (starting from 0) when you click and place a block or xref. This option ignores the angle specified under Rotation in the Tool Properties dialog box. The rotation angle prompt does not display if you drag the block or xref, or if you enter rotate at the initial insertion Command prompt.

Blocks that are placed by dragging from a tool palette must often be rotated or scaled after placement. You can use object snaps when dragging blocks from a tool palette; however, grid snap is suppressed during dragging. You can set an auxiliary scale for a block or a hatch tool to override the regular scale setting when you use the tool. (An auxiliary scale multiplies your current scale setting by the plot scale or the dimension scale.)

Blocks dragged from a tool palette are automatically scaled according to the ratio of units in both the block and the current drawing. For example, if the current drawing uses meters as units and a block uses centimeters, the unit ratio is 1 m/100 cm. When you drag the block into the drawing, it is inserted at 1/100 scale.

NOTE In the Options dialog box, User Preferences tab, the Source Content Units and Target Drawing Units settings are used when Drag-and-Drop Scale is set to Unitless, either in the source block or target drawing.

Update Block Definitions on Tool Palettes

A block definition in your current drawing does not update automatically when you modify the block in the source drawing. To update a block definition in the current drawing, right-click the block tool on the tool palette and click Redefine on the shortcut menu.

If the Redefine option is unavailable, then the block definition source is a drawing file rather than a block within a drawing file. To update a block definition that was created by inserting a drawing file, use DesignCenter. For more information, see Add Content with DesignCenter on page 96.

NOTE If you move the source drawing file for a block tool to a different folder, then modify the tool that references it by right-clicking the tool and, in the Tool Properties dialog box, specifying the new source file folder.

See also:

- Control Tool Properties on page 70
- Customize Tool Palettes on page 74
- Create Task-Based Workspaces on page 151
- Add Content with DesignCenter on page 96

To display the Tool Palettes window

Do one of the following:

- Click View tab ➤ Palettes panel ➤ Tool Palettes.
- Press Ctrl+3.

Command entry: TOOLPALETTES

Toolbar: Standard

To create a tool from an object in the current drawing

1. In the current drawing, select an object such as a dimension, block, hatch, gradient fill, raster image, xref, or any geometric object.
2. Drag the object to a tool palette and, without releasing the mouse button, move the cursor to the place on the tool palette where you want the tool.

   You can switch to a different tab by hovering over the tab for a few seconds. The black line indicates where the tool is located.
3. Release the mouse button.

Command entry: TOOLPALETTES

Toolbar: Standard

To use a tool created from a geometric object

1. On a tool palette, click the geometric object tool you want to use.
2. Use the tool as if you selected the corresponding option from the Draw menu or the corresponding button on the Draw toolbar, following the Command prompts.

   The geometric object you create has the same properties as the tool you selected from the tool palette.

Command entry: TOOLPALETTES

Toolbar: Standard

To use a tool created from a dimension

1. On a tool palette, click the dimension tool you want to use.
2. Use the tool as if you selected the corresponding option from the Dimension menu or the corresponding button on the Dimension toolbar, following the Command prompts.

   The dimension you create has the same dimension style and properties as the tool you selected from the tool palette.

Toolbar: Standard
Command entry: TOOLPALETTES

To use a tool on a tool flyout

1. On a tool palette, click the arrow on the right side of the dimension tool or the geometric object tool you want to use.
2. On the flyout, select a tool.
3. Use the tool as if you selected the corresponding option from a menu or the corresponding button on a toolbar, following the Command prompts.

   The object you create has the same properties as the tool whose icon you selected on the tool flyout.

Toolbar: Standard
Command entry: TOOLPALETTES

To add or remove a tool flyout

1. On a tool palette, right-click the geometric object tool or the dimension tool whose flyout you want to add or remove. Click Properties.
2. In the Tool Properties dialog box, under Command, click in the Use Flyout box.
3. In the drop-down list, select Yes if you want to add a flyout, or select No if you want to remove one.
4. Click OK.

NOTE If you remove the flyout from a tool but then add the flyout back, the image, name, and description (the tooltip) that displays on the tool palette for each tool on the flyout will not be accurate. To correct this, return the image, name, and description of the flyout tool to the default settings. See To change the image, name, and description of a flyout tool to the default settings.

Toolbar: Standard
Command entry: TOOLPALETTES

To customize a tool flyout

1. On a tool palette, right-click the geometric object tool or dimension tool whose flyout you want to customize. Click Properties.
2. In the Tool Properties dialog box, under Command, click in the Flyout Options box. Click the [...] button.
3. In the Flyout Options dialog box, select the tools that you want to show on the flyout. Click OK. (You must select at least one tool.)
4. In the Tool Properties dialog box, click OK.

Toolbar: Standard
Command entry: TOOLPALETTES

To use a tool created from a hatch

1. On a tool palette, click a hatch tool and drag it to an object in the drawing.
2. Release the mouse button to apply the hatch to the object.

   The hatch you create has the same hatch style and properties as the tool you selected from the tool palette.

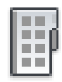
Toolbar: Standard
Command entry: TOOLPALETTES

To use a tool created from a gradient fill

1 On a tool palette, click a gradient fill tool and drag it to an object in the drawing.

2 Release the mouse button to apply the gradient fill to the object.

The gradient fill you create has the same style and properties as the tool you selected from the tool palette.

Toolbar: Standard 
Command entry: TOOLPALETTES

To use a tool created from a block, xref, or raster image

1 On a tool palette, click the block, xref, or raster image you want to insert into your drawing.

2 Drag the block, xref, or raster image to the appropriate place in the drawing.

3 Release the mouse button to insert the block, xref, or raster image.

The block, xref, or raster image you insert has the same properties as the tool you selected from the tool palette.

Toolbar: Standard 
Command entry: TOOLPALETTES

To be prompted for a rotation angle when placing a block or xref from a tool palette

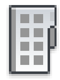
1 In a tool palette, right-click a block or xref tool. Click Properties.

2 In the Tool Properties dialog box, under Insert, click Prompt for Rotation.

3 In the drop-down list, select Yes.

4 Click OK.

NOTE This option ignores the angle specified under Rotation in the Tool Properties dialog box. The rotation angle prompt does not display if you drag the block or xref, or if you enter **rotate** at the initial insertion Command prompt.

Toolbar: Standard 
Command entry: TOOLPALETTES

Quick Reference

Commands

CUSTOMIZE

Customizes tool palettes and tool palette groups.

TOOLPALETTES

Opens the Tool Palettes window.

TOOLPALETTESCLOSE

Closes the Tool Palettes window.

UNITS

Controls coordinate and angle display formats and precision.

System Variables

INSUNITSDEFSOURCE

Sets source content units value when INSUNITS is set to 0.

INSUNITSDEFTARGET

Sets target drawing units value when INSUNITS is set to 0.

PALETTEOPAQUE

Controls whether palettes can be made transparent.

TPSTATE

Indicates whether the Tool Palettes window is open or closed.

Utilities

No entries

Command Modifiers

No entries

Create and Use Command Tools

You can create a tool on a tool palette that executes a single command or a string of commands.

You can add frequently used commands to a tool palette. When the Customize dialog box is open, you can drag tools from a toolbar to a tool palette or you can drag tools from the Customize User Interface (CUI) Editor to a tool palette.

Once you add a command to a tool palette, you can click the tool to execute the command. For example, clicking a Save tool on a tool palette saves a drawing just as the Save button on the Standard toolbar does.

You can also create a tool that executes a string of commands or customized commands, such as an AutoLISP® routine, a VBA macro or application, or a script.

NOTE Even though the tools on palettes can be clicked when the Customize User Interface (CUI) Editor is displayed, the end results might be unpredictable. It is best to not use any of the tools on a palette while the Customize User Interface (CUI) Editor is displayed.

To create a command tool from a toolbar button

1. Make sure the toolbar that contains the command you want to add to the tool palette is displayed.

   If the required toolbar is not displayed, click Tools ➤ Toolbars and select another toolbar from the list.

2. Click Manage tab ➤ Customization panel ➤ Tool Palettes. 

   NOTE Even though you won't make any changes in the Customize dialog box in this procedure, it must be displayed when you add command tools to a tool palette.

3 In the program, drag a command (button) from a toolbar to the tool palette and, without releasing the mouse button, move the cursor to the place on the tool palette where you want the tool.

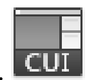
   The black horizontal line indicates where the tool will be located.

4 Release the mouse button.

5 In the Customize dialog box, click Close.

Command entry: CUSTOMIZE

To create a command tool from the Customize User Interface

1 Click Manage tab ➤ Customization panel ➤ User Interface. 

   If the CUI Editor is covering the Tool Palettes window, then move the CUI Editor to the side.

2 In the Command List pane, drag a command to the tool palette and, without releasing the mouse button, move the cursor to the place on the tool palette where you want the tool.

   The black horizontal line indicates where the tool will be located.

3 Release the mouse button.

4 In the CUI Editor, click Close.

Command entry: CUI

To create a command tool that executes multiple or customized commands (advanced)

1 Click Manage tab ➤ Customization panel ➤ Tool Palettes. 

2 In the program, drag a command from a toolbar to the tool palette and, without releasing the mouse button, move the cursor to the place on the tool palette where you want the tool.

3 Release the mouse button.

4 On the tool palette, right-click the tool. Click Properties.

5 In the Tool Properties dialog box, change the name and description to an appropriate name and description for the string, AutoLISP routine, or script.

6 Under Command, in the Command String box, enter a string of commands or customized commands, such as an AutoLISP routine, a VBA macro or application, or a script.

7 Click OK.

Command entry: CUSTOMIZE

To use a command tool

1 On a tool palette, click the command tool that you want to use.

2 Follow any Command prompts that are shown.

Quick Reference

Commands

CUI

Manages the customized user interface elements in the product.

CUSTOMIZE

Customizes tool palettes and tool palette groups.

TOOLPALETTES

Opens the Tool Palettes window.

TOOLPALETTESCLOSE

Closes the Tool Palettes window.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Change Tool Palette Settings

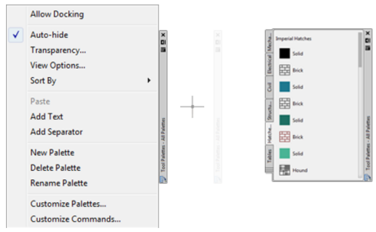
The options and settings for tool palettes are accessible from shortcut menus that are displayed when you right-click in different areas of the Tool Palettes window.

You can dock the Tool Palettes window on the right or left edge of the application window. Press the Ctrl key if you want to prevent docking as you move the Tool Palettes window.

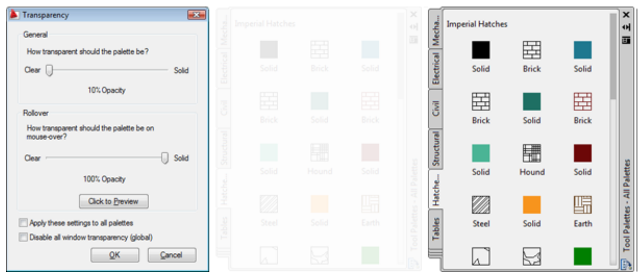
Tool palette settings are saved with your profile.

These settings include

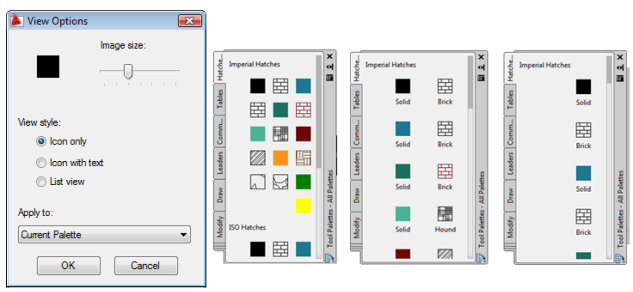
- **Allow Docking.** Toggles the ability to dock or anchor palette windows. If this option is selected, a window can be docked when you drag it over a docking area at the side of a drawing. A docked window adheres to the side of the application window and causes the drawing area to be resized. Selecting this option also makes Anchor Right and Anchor Left available.
- **Anchor Left** or **Anchor Right**. Attaches the palette to an anchor tab base at the left or right 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.** Controls the display of the palette when it is floating. When this option is selected, only the tool palette title bar is displayed when the cursor moves outside the tool palette. When this option is cleared, the palette stays open continuously. You can display the tool palette title bar as either icons or text from the shortcut menu of the title bar.



- **Transparency.** Sets the transparency of the Tool Palettes window so it does not obscure objects under it.



- **Views.** Changes the display style and size of the icons in a tool palette.



To change the rollover behavior of the Tool Palettes window

- In the Tool Palettes window, at the top of the title bar, click the Auto-Hide button.

NOTE Rollover behavior is available only when the Tool Palettes window is undocked.

To change the transparency of the Tool Palettes window

1. In the Tool Palettes window, at the top of the title bar, click the Properties button. Click Transparency.
2. In the Transparency dialog box, adjust the level of transparency for the Tool Palettes window. Click OK.

NOTE Transparency is available only when the Tool Palettes window is undocked.

To change the icon display style in the Tool Palettes window

1. Right-click a blank area inside the Tool Palettes window. Click View Options.
2. In the View Options dialog box, click the icon display option that you want to set. You can also change the size of the icons.
3. Click the list box under Apply To, and then select either Current Tool Palette or All Tool Palettes.
4. Click OK.

Quick Reference

Commands

CUSTOMIZE

Customizes tool palettes and tool palette groups.

TOOLPALETTES

Opens the Tool Palettes window.

TOOLPALETTESCLOSE

Closes the Tool Palettes window.

System Variables

PALETTEOPAQUE

Controls whether palettes can be made transparent.

TPSTATE

Indicates whether the Tool Palettes window is open or closed.

Utilities

No entries

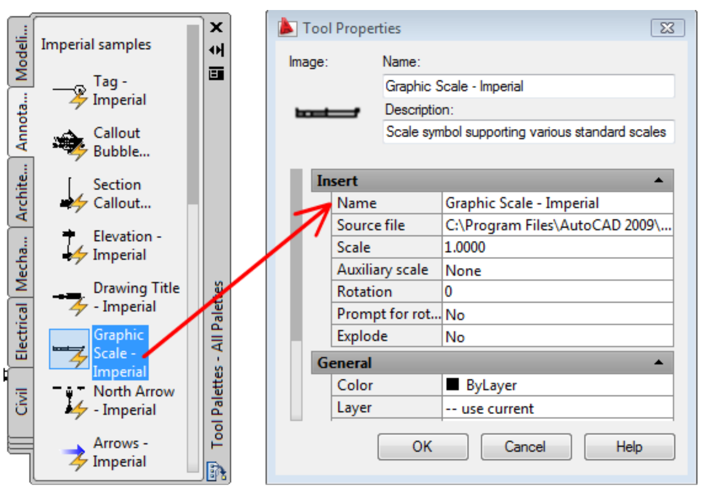
Command Modifiers

No entries

Control Tool Properties

You can change the properties of any tool on a tool palette.

Once a tool is on a tool palette, you can change its properties. For example, you can change the insertion scale of a block or the angle of a hatch pattern.



To change tool properties, right-click on a tool, and then click Properties on the shortcut menu to display the Tool Properties dialog box. The Tool Properties dialog box has the following two categories of properties:

- **Insert or Pattern properties.** Control object-specific properties such as scale, rotation, and angle.
- **General properties.** Override the current drawing property settings such as layer, color, and linetype.

You can expand and collapse the property categories by clicking the arrow buttons.

Specify a Different Icon for a Tool

You can replace the icon for a tool with an image that you specify. This is useful when the automatically generated icon is too cluttered to be easily recognizable.

To replace the image, right-click the tool in the tool palette and click Specify Image on the shortcut menu.

To restore the default image for a tool, right-click the tool and click Remove Specified Image.

Update the Icon for a Tool

The icon for a block, xref, or raster image in a tool palette is not automatically updated if its definition changes. If you change the definition for a block, xref, or raster image, you can update the icon by right-clicking the tool in the palette and clicking Update Tool Image. You must save the drawing before you can update the tool image.

Alternatively, you can delete the tool, and then replace it using DesignCenter™.

Specify Overrides for Tool Properties

In some cases, you may want to assign specific property overrides to a tool. For example, you may want a hatch to be placed automatically on a pre-specified layer, regardless of the current layer setting. This feature can save you time and reduce errors by setting properties automatically when creating certain objects.

The Tool Properties dialog box provides areas for each possible property override.

Layer property overrides affect color, linetype, lineweight, plot style, and plot. Layer property overrides are resolved as follows:

- If a layer is missing from the drawing, that layer is created automatically.
- If a layer to which you are adding content is currently turned off or frozen, the layer is temporarily turned on or thawed.

To display the properties of a tool on a tool palette

1 On a tool palette, right-click a tool. Click Properties.

2 In the Tool Properties dialog box, use the scroll bar to view all tool properties.

   You can resize the Tool Properties dialog box by dragging an edge, or you can expand and collapse the property categories by clicking the double arrow buttons.

3 Click OK.

To change the property of a tool on a tool palette

1 On a tool palette, right-click a tool. Click Properties.

2 In the Tool Properties dialog box, click any property in the list of properties and specify the new value or setting.

   - Properties listed under the Insert or Pattern category control object-specific properties such as scale, rotation, and angle.
   - Properties listed under the General category override the current drawing property settings such as layer, color, and linetype.
   - Auxiliary scale for a block or a hatch tool overrides the regular scale setting when the tool is used. (An auxiliary scale multiplies your current scale setting by the plot scale or the dimension scale.)

   You can resize the Tool Properties dialog box by dragging an edge, or you can expand and collapse the property categories by clicking the arrow buttons.

3 Click OK.

NOTE If you specify an image, name, or description for a tool that has a flyout, that image, name, and description are displayed on the tool palette for each tool on the flyout. To return the flyout tool's image, name, and description to their default settings, leave the corresponding boxes blank in the Tool Properties dialog box.

Toolbar: Standard 
Command entry: TOOLPALETTES

To refresh the image of a block tool on a tool palette

1 On a tool palette, right-click a tool.

2 Click Update Tool Image.

NOTE You must save the drawing before you can update the tool image.

Toolbar: Standard
Command entry: TOOLPALETTES

To change the image of a tool on a tool palette

1 On a tool palette, right-click a tool.

2 On the shortcut menu, click Specify Image.

3 In the Select Image File dialog box, locate the image file you want to use.

4 Click Open to insert the new image.

NOTE If you specify an image for a tool that has a flyout, that image is displayed on the tool palette for each tool on the flyout. To return the flyout tool images to their default images, see To change the image, name, and description of a flyout tool to the default settings.

Toolbar: Standard
Command entry: TOOLPALETTES

To change the image, name, and description of a flyout tool to the default settings

1 On a tool palette, right-click a tool. Click Properties.

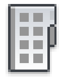
2 In the Tool Properties dialog box, right-click in the image area. Click Delete Image.

3 Click in the Name box and delete the text.

4 Click in the Description box and delete the text.

5 Click OK.

NOTE Leaving the Image, Name, and Description boxes blank in the Tool Properties dialog box will return the flyout tool's image, name, and description to their default settings.

Toolbar: Standard 
Command entry: TOOLPALETTES

Quick Reference

Commands

CUSTOMIZE

Customizes tool palettes and tool palette groups.

TOOLPALETTES

Opens the Tool Palettes window.

TOOLPALETTESCLOSE

Closes the Tool Palettes window.

TPNAVIGATE

Displays a specified tool palette or palette group.

System Variables

PALETTEOPAQUE

Controls whether palettes can be made transparent.

TPSTATE

Indicates whether the Tool Palettes window is open or closed.

Utilities

No entries

Command Modifiers

No entries

Customize Tool Palettes

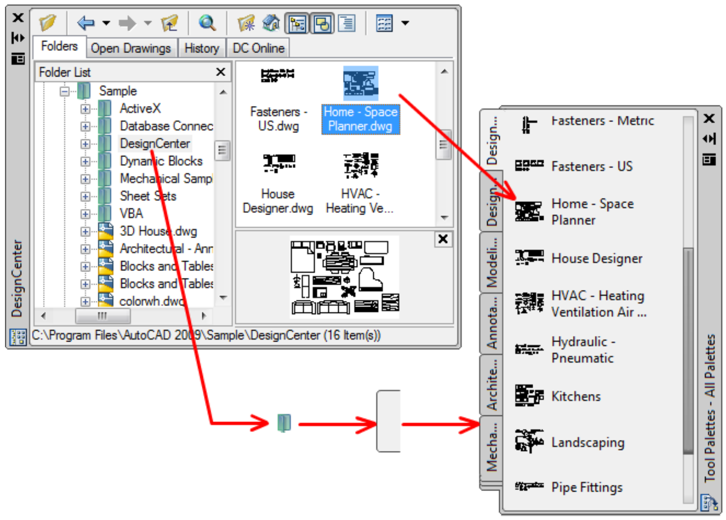
You can add tools to a tool palette with several methods.

You can create new tool palettes using the Properties button on the title bar of the Tool Palettes window. Add tools to a tool palette with the following methods:

- Drag any of the following onto your tool palette: geometric objects such as lines, circles, and polylines; dimensions; hatches; gradient fills; blocks; xrefs; raster images.
- Drag drawings, blocks, and hatches from DesignCenter to the tool palette. Drawings that are added to a tool palette are inserted as blocks when dragged into the drawing.
- Use the Customize dialog box to drag commands to a tool palette just as you might add them to a toolbar.
- Use the Customize User Interface (CUI) Editor to drag commands to a tool palette from the Command List pane.
- Use Cut, Copy, and Paste to move or copy tools from one tool palette to another.
- Manage tool palettes by creating new palettes from scratch, renaming, deleting or moving palettes with the shortcut menu.

  NOTE It is not recommended to create or rename tool palettes when the Customize User Interface (CUI) Editor is displayed.

- Create a tool palette tab with predetermined content by right-clicking a folder, a drawing file, or a block in the DesignCenter tree view, and then clicking Create Tool Palette on the shortcut menu.



- You can associate a customizable tool palette group with each panel on the ribbon. Right-click the ribbon panel to display a list of available tool palette groups.

NOTE If the source drawing file for a block, xref, or raster image tool is moved to a different folder, you must modify the tool that references it by right-clicking the tool and, in the Tool Properties dialog box, specifying the new source file folder.

Rearranging Tools and Tool Palettes

Once tools are placed on a tool palette, you can rearrange them by dragging them around or by sorting them. You can also add text and separator lines to tool palettes.

You can move a tool palette tab up and down the list of tabs by using the tool palette shortcut menu or the Tool Palettes tab of the Customize dialog box. Similarly, you can delete tool palettes that you no longer need. Tool palettes that are deleted are lost unless they are first saved by exporting them to a file. You can control the path to your tool palettes on the Files tab in the Options dialog box. This path can be to a shared network location.

Read-Only Tool Palettes

If a tool palette file is set with a read-only attribute, a lock icon is displayed in a lower corner of the tool palette. This indicates that you cannot modify the tool palette beyond changing its display settings and rearranging the icons.

To apply a read-only attribute to a tool palette, right-click the tool palette (ATC) file in the following location: *C:\documents and settings\<user name>\application data\autodesk\AutoCAD 2010\r17.2\enu\support\ToolPalette\Palettes*. On the shortcut menu, click Properties. On the General tab, select Read-only, and click OK.

To create a tool palette

1. Click View tab ➤ Palettes panel ➤ Tool Palettes.
2. In the Tool Palettes window, at the top of the title bar, click the Properties button. Click New Palette.
3. In the text box, enter a name for the new palette.
4. If necessary, right-click over the tab and select Move Up or Move Down to change the order in which the tab appears in.

Command entry: TOOLPALETTES

Toolbar: Standard

To create a tool palette from a folder or a drawing

1. If DesignCenter is not already open, click View tab ➤ Palettes panel ➤ DesignCenter.
2. In the DesignCenter tree view or the content area, right-click a folder, drawing file, or block. Click Create Tool Palette.

   A new tool palette is created that contains all the blocks and hatches in the selected folder or drawing.

Command entry: ADCENTER

Toolbar: Standard

To associate a tool palette group with a ribbon panel

1. Click View tab ➤ Palettes panel ➤ Tool Palettes.

2 On the ribbon, right-click a ribbon panel and click Tool Palette Group.

3 Click an available tool palette group from the list.

4 Click Tools menu ➤ Workspaces ➤ Save Current As.

5 In the Save Workspace dialog box, enter a name for the new workspace or select a name from the drop-down list. Click Save.

To display the tool palette group associated with a ribbon panel

- Right-click a ribbon panel and click Show Related Tool Palette Group.

To rename a tool palette

1 Right-click a blank area inside the Tool Palettes window. Click Rename Palette.

2 In the text box, enter a new name for the palette.

Command entry: TOOLPALETTES

Toolbar: Standard 

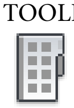
To delete a tool palette

1 Right-click a blank area inside the Tool Palettes window. Click Delete Palette.

2 In the Confirm Palette Deletion message box, click OK to delete the tool palette.

NOTE There is no way to undo the deletion of a tool palette. It is recommended that you first export the tool palette before making any deletions. Use the Export option in the Customize dialog box.

Command entry: TOOLPALETTES

Toolbar: Standard 

To change the position of a tool palette

1 Right-click the tab that represents the tool palette you want to move in the Tool Palettes Window. Click Move Up or Move Down.

2 Repeat step 1 until the tool palette has been moved to the desired location.

NOTE You can also change the order of a tool palette by right-clicking on the title bar of a tool palette, and then clicking Customize Palettes.

Command entry: TOOLPALETTES

Toolbar: Standard

To add text to a tool palette

1 Right-click a blank area inside the Tool Palettes window. Click Add Text.

2 In the text box, add the text you want to display in the window.

3 If necessary, drag the text to the appropriate location in the window.

Command entry: TOOLPALETTES

Toolbar: Standard

To add a separator line to a tool palette

1 Right-click a blank area inside the Tool Palettes window. Click Add Separator.

2 If necessary, drag the separator to the appropriate location in the window.

Command entry: TOOLPALETTES

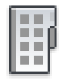
Toolbar: Standard

To sort items in a tool palette

- Right-click a blank area inside the Tool Palettes window. Click Sort By ➤ Name or Sort By ➤ Type.

NOTE Items are sorted in order, by text, separator, and tool.

Command entry: TOOLPALETTES

Toolbar: Standard 

Quick Reference

Commands

CUSTOMIZE

Customizes tool palettes and tool palette groups.

TOOLPALETTES

Opens the Tool Palettes window.

TOOLPALETTESCLOSE

Closes the Tool Palettes window.

System Variables

PALETTEOPAQUE

Controls whether palettes can be made transparent.

TPSTATE

Indicates whether the Tool Palettes window is open or closed.

Utilities

No entries

Command Modifiers

No entries

Organize Tool Palettes

You can organize tool palettes into groups and specify which group of tool palettes is displayed.

For example, if you have several tool palettes that contain hatch patterns, you can use the CUSTOMIZE command to create a new palette group called Hatch Patterns. You can then add all your tool palettes that contain hatch patterns to the Hatch Pattern group.

When you set the Hatch Pattern group as the current group, only those tool palettes you've added to the group are displayed.

To create a tool palette group

1. Click Manage tab ➤ Customization panel ➤ Tool Palettes.
2. In the Customize dialog box, under Palette Groups, right-click on the lower, blank area. Click New Group.

   If there are no groups listed in the Palette Groups area, you can create a group by dragging a tool palette from the Tool Palettes area into the Palette Groups area.
3. Enter a name for the tool palette group.
4. Click Close.

Command entry: CUSTOMIZE

To add a tool palette to a tool palette group

1. Click Manage tab ➤ Customization panel ➤ Tool Palettes.
2. In the Customize dialog box, drag a tool palette from the Tool Palettes area into a group in the Palette Groups area.
3. Click Close.

Command entry: CUSTOMIZE

To remove a tool palette from a tool palette group

1. Click Manage tab ➤ Customization panel ➤ Tool Palettes.
2. In the Customize dialog box, under Palette Groups, right-click the name of the tool palette you want to remove. Click Remove.

   You can also drag the tool palette into the Tool Palettes area to remove it from a group.
3. Click Close.

Command entry: CUSTOMIZE

To display a tool palette group

1 Right-click on the title bar of a tool palette.

2 Click the name of the tool palette group that you want to display.

Command entry: CUSTOMIZE, TPNAVIGATE

To delete a tool palette group

1 Click Manage tab ➤ Customization panel ➤ Tool Palettes. 

2 In the Customize dialog box, under Palette Groups, right-click the tool palette group you want to delete. Click Delete.

NOTE You cannot delete a tool palette group if it is set as the current group. To delete a group that is set as the current group, you must first set another group as the current group.

3 Click Close.

Command entry: CUSTOMIZE

To delete all tool palette groups

1 Right-click on the title bar of a tool palette. Click All Palettes.

NOTE You must display all tool palettes so that no tool palette group is set as the current group.

2 Click Manage tab ➤ Customization panel ➤ Tool Palettes. 

3 In the Customize dialog box, under Palette Groups, right-click a tool palette group. Click Delete.

4 Repeat step 3 until all tool palette groups are deleted.

5 Click Close.

Command entry: CUSTOMIZE

To rename a tool palette group

1 Click Manage tab ➤ Customization panel ➤ Tool Palettes. 

2 In the Customize dialog box, under Palette Groups, right-click the tool palette group you want to rename. Click Rename.

3 Enter a new name for the tool palette group.

4 Click Close.

Command entry: CUSTOMIZE

To rearrange tool palette groups

1 Click Manage tab ➤ Customization panel ➤ Tool Palettes. 

2 In the Customize dialog box, under Palette Groups, click a tool palette group and drag it into the new position.

   Any other tool palette groups contained in the group you move will also be moved.

3 Click Close.

NOTE You cannot drag a tool palette group into a group that is contained by it.

Command entry: CUSTOMIZE

To create a tool palette group within another one

1 Click Manage tab ➤ Customization panel ➤ Tool Palettes. 

2 In the Customize dialog box, under Palette Groups, right-click the tool palette group to which you want to add the new group. Click New Group.

3 Enter a name for the new tool palette group.

4 Click Close.

Command entry: CUSTOMIZE

To copy and paste a tool palette from one group to another

1 Click Manage tab ➤ Customization panel ➤ Tool Palettes.

2 In the Customize dialog box, under Palette Groups, select the tool palette that you want to copy.

3 Press Ctrl while you drag the selected tool palette to another group.

   When you release the mouse button, a copy of the tool palette is displayed in the new location.

4 Click Close.

Command entry: CUSTOMIZE

To change the order of displayed tool palettes within a group

1 Click Manage tab ➤ Customization panel ➤ Tool Palettes.

2 In the Customize dialog box, under Palette Groups, drag a tool palette to the new location in its tool palette group.

3 Click Close.

Command entry: CUSTOMIZE

To change the order of displayed tool palettes when all of them are displayed

1 Click Manage tab ➤ Customization panel ➤ Tool Palettes.

2 In the Customize dialog box, under Tool Palettes, drag a tool palette to the location you want.

   When all tool palettes are displayed, they are shown in the order they appear in this list.

3 Click Close.

Command entry: CUSTOMIZE

To display all tool palettes

- Right-click on the title bar of a tool palette. Click All Palettes.

Quick Reference

Commands

CUSTOMIZE

Customizes tool palettes and tool palette groups.

TOOLPALETTES

Opens the Tool Palettes window.

TOOLPALETTESCLOSE

Closes the Tool Palettes window.

TPNAVIGATE

Displays a specified tool palette or palette group.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Save and Share Tool Palettes

You can save and share a tool palette or tool palette group by exporting it or importing it as a file.

You can save and share a tool palette by exporting it or importing it as a tool palette file. Tool palette files have an *.xtp* file extension.

Similarly, you can save and share a tool palette group by exporting it or importing it as a palette group file. Tool palette files have an *.xpg* file extension.

In some cases, when you export a customized tool palette, an image folder with the same name as the exported tool palette is automatically created in the same location as the XTP file. This image folder contains the icon images used on the exported tool palette. The folder is created when you export a tool palette that contains any of the following items:

- User-created content tools
- Command tools that contain user-specified (custom) tool palette icons (images)

When you import a customized tool palette, this image folder must be in the same location as the imported XTP file in order for the icons to appear on the tool palette.

Tool palettes can be used only in the version of AutoCAD in which they were created. For example, you cannot use a tool palette that was created in AutoCAD 2010 in AutoCAD 2005.

The default path for tool palette files is set on the Files tab of the Options dialog box under Tool Palettes File Locations.

Tool palette groups are saved in profiles.

If you send tool palettes to someone who uses AutoCAD LT, note that some tools created in AutoCAD do not behave the same way or work in AutoCAD LT. Note the following limitations:

- The color property of tools that use a color other than an AutoCAD Color Index (ACI) color convert to ByLayer in AutoCAD LT.
- Gradient fill tools switch to hatch tools in AutoCAD LT.
- Raster image tools do not work in AutoCAD LT.

NOTE If a tool palette file is set with a read-only attribute, a lock icon is displayed in a lower corner of the tool palette. This indicates that you cannot modify the tool palette beyond changing its display settings and rearranging the icons.

See also:

- Save and Restore Interface Settings (Profiles) on page 155

To share a tool palette

1 Click Manage tab ➤ Customization panel ➤ Tool Palettes. 

2 In the Customize dialog box, under Palettes, right-click a tool palette. On the shortcut menu, click Export.

3 In the Export Palette dialog box, enter a file name and click Save.

4 Click Close.

Command entry: CUSTOMIZE

To share a tool palette group

1 Click Manage tab ➤ Customization panel ➤ Tool Palettes. 

2 In the Customize dialog box, under Palette Groups, right-click a tool palette group. Click Export.

3 In the Export Group dialog box, enter a file name and click Save.

4 Click Close.

Command entry: CUSTOMIZE

Quick Reference

Commands

CUSTOMIZE

Customizes tool palettes and tool palette groups.

TOOLPALETTES

Opens the Tool Palettes window.

TOOLPALETTESCLOSE

Closes the Tool Palettes window.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

DesignCenter

With DesignCenter, you can organize access to blocks, hatches, xrefs, and other drawing content. You can drag content from any source drawing to your current drawing. You can drag drawings, blocks, and hatches to a tool palette. Source drawings can be on your computer, on a network location, or on a website. In addition, if you have multiple drawings open, you can use DesignCenter to streamline your drawing process by copying and pasting other content, such as layer definitions, layouts, and text styles between drawings.

Overview of DesignCenter

With DesignCenter, you can

- Browse for drawing content such as drawings or symbol libraries on your computer, on a networked drive, and on a web page
- View definition tables for named objects such as blocks and layers in any drawing file and then insert, attach, or copy and paste the definitions into the current drawing
- Update (redefine) a block definition
- Create shortcuts to drawings, folders, and Internet locations that you access frequently
- Add content such as xrefs, blocks, and hatches to a drawing
- Open drawing files in a new window
- Drag drawings, blocks, and hatches to a tool palette for convenient access

Quick Reference

Commands

ADCENTER

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

ADCNAVIGATE

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

System Variables

ADCSTATE

Indicates whether the DesignCenter window is open or closed.

Utilities

No entries

Command Modifiers

No entries

Understand the DesignCenter Window

You can control the size, location, and appearance of DesignCenter.

The Organization of the DesignCenter Window

The DesignCenter window is divided into the tree view on the left side and the content area on the right side. Use the tree view to browse sources of content and to display content in the content area. Use the content area to add items to a drawing or to a tool palette.

Undocked, the DesignCenter window is displayed as shown.

Below the content area, you can also display a preview or a description of a selected drawing, block, hatch pattern, or xref. A toolbar at the top of the window provides several options and operations.

Control the Size, Location, and Appearance of DesignCenter

You can control the size, location, and appearance of DesignCenter. Many of these options can be set by right-clicking and selecting an option on the shortcut menu.

- Resize DesignCenter by dragging the bar between the content area and the tree view or by dragging an edge of the window.
- Dock DesignCenter by dragging it over the right or left docking region of the application window until it snaps into the docked position. You can also dock the DesignCenter window by double-clicking its title bar.
- Undock DesignCenter by dragging the area above the toolbar away from the docking region. Pressing Ctrl while dragging prevents docking.
- Anchor DesignCenter by choosing Anchor Right or Anchor Left from the shortcut menu. An anchored DesignCenter window rolls open and closed as the cursor moves across it. When an anchored DesignCenter window is open, its content overlaps the drawing area. It cannot be set to stay open.
- When DesignCenter is floating, use Auto-hide to set it to roll open and closed as the cursor moves across it.

The DesignCenter Toolbar

The DesignCenter toolbar controls navigation and display of information in the tree view and the content area. For information about these buttons, see the ADCENTER command. The same navigation and display options are available on the shortcut menu. Right-click in the DesignCenter content area to display the menu.

To change the DesignCenter rollover behavior

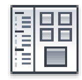
1. Click View tab ➤ Palettes panel ➤ DesignCenter. 
2. Right-click the DesignCenter title bar. Click Auto-hide.

   When the DesignCenter rollover option is turned on, the DesignCenter tree view and content area disappear when you move your cursor off the DesignCenter window, leaving only the title bar. When you move your cursor over the title bar, the DesignCenter window is restored.

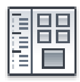
Command entry: ADCENTER

Toolbar: Standard 

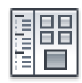
To prevent DesignCenter from docking

1. Click View tab ➤ Palettes panel ➤ DesignCenter. 
2. On the DesignCenter title bar, click Propert. Press Ctrl as you move your mouse.

Command entry: ADCENTER

Toolbar: Standard 

To display and hide the DesignCenter tree view

1. Click View tab ➤ Palettes panel ➤ DesignCenter. 
2. On the DesignCenter toolbar, click Tree View Toggle.

Command entry: ADCENTER

Toolbar: Standard 

Quick Reference

Commands

ADCENTER

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

ADCNAVIGATE

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

System Variables

ADCSTATE

Indicates whether the DesignCenter window is open or closed.

Utilities

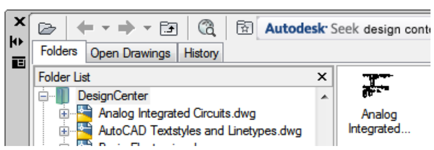
No entries

Command Modifiers

No entries

Access Content with DesignCenter

The tree view in the left portion of the DesignCenter window and the four DesignCenter tabs help you find and load content into the content area.



Folders Tab

The Folders tab displays a hierarchy of navigational icons, including

- Networks and computers
- Web addresses (URLs)
- Computer drives
- Folders
- Drawings and related support files
- Xrefs, layouts, hatch styles, and named objects, including blocks, layers, linetypes, text styles, dimension styles, table styles, multileader styles, and plot styles within a drawing

Click an item in the tree view to display its contents in the content area. Click the plus (+) or minus (-) sign to display and hide additional levels in the hierarchy. You can also double-click an item to display deeper levels. Right-clicking in the tree view displays a shortcut menu with several related options.

Open Drawings, History, and DC Online Tabs

The Open Drawings, History, and DC Online tabs provide alternate methods of locating content.

- **Open Drawings.** Displays a list of the drawings that are currently open. Click a drawing file and then click one of the definition tables from the list to load the content into the content area.
- **History.** Displays a list of files opened previously with DesignCenter. Double-click a drawing file from the list to navigate to the drawing file in the tree view of the Folders tab and to load the content into the content area.
- **DC Online.** Provides content from the DesignCenter Online web page including blocks, symbol libraries, manufacturer's content, and online catalogs.

NOTE The DesignCenter Online (DC Online tab) is disabled by default. You can enable it from the CAD Manager Control utility on page 101.

Bookmark Frequently Used Content

DesignCenter provides a solution to finding content that you need to access quickly on a regular basis. Both the tree view and the content area include options that activate a folder called *Favorites*. The *Favorites* folder can contain shortcuts to content on local or network drives as well as in Internet locations.

When you select a drawing, folder, or another type of content and choose Add to Favorites, a shortcut to that item is added to the *Favorites* folder. The original file or folder doesn't actually move; in fact, all the shortcuts you create are stored in the *Favorites* folder. The shortcuts saved in the *Favorites* folder can be moved, copied, or deleted using Windows® Explorer.

To change the source of the content displayed in DesignCenter

1. Click View tab ➤ Palettes panel ➤ DesignCenter. 
2. On the DesignCenter window, click one of the following tabs:
   - **Folders**. Lists your local and network drives.
   - **Open Drawings**. Lists the drawings that are currently open.
   - **History**. Lists the last 20 locations accessed through DesignCenter.

■ **DC Online**. Displays online content from the Web.

NOTE The DesignCenter Online (DC Online tab) is disabled by default. You can enable it from the CAD Manager Control utility on page 101.

Command entry: ADCENTER

Toolbar: Standard

To change the folder of the Home button in DesignCenter

1 Click View tab ➤ Palettes panel ➤ DesignCenter.

2 In the DesignCenter tree view, navigate to the folder that you want to set as home.

3 Right-click on the folder. Click Set as Home.

When you click the Home button, DesignCenter will automatically load this folder.

Command entry: ADCENTER

Toolbar: Standard

To add items to the Favorites folder in DesignCenter

1 Click View tab ➤ Palettes panel ➤ DesignCenter.

2 Right-click the item in the DesignCenter tree view or content area. Click Add to Favorites.

Toolbar: Standard

Command entry: ADCENTER

To display the contents of the Favorites folder in DesignCenter

1 Click View tab ➤ Palettes panel ➤ DesignCenter.

2 In DesignCenter, click the Favorites button.

When you are working in the tree view, you can use the Folders tab to navigate to the *Favorites* folder.

Command entry: ADCENTER

Toolbar: Standard

To organize your DesignCenter Favorites folder

1 Click View tab ➤ Palettes panel ➤ DesignCenter.

2 In DesignCenter, click the Favorites button.

3 Right-click the background in the content area. Click Organize Favorites.

Your Autodesk *Favorites* folder is opened in a window.

Toolbar: Standard

Command entry: ADCENTER

Quick Reference

Commands

ADCENTER

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

ADCNAVIGATE

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

System Variables

ADCSTATE

Indicates whether the DesignCenter window is open or closed.

Utilities

No entries

Command Modifiers

No entries

Add Content with DesignCenter

The right portion of the DesignCenter window operates on the content displayed.

Double-clicking an item in the content area displays successive levels of detail. For example, double-clicking a drawing image displays several icons, including an icon for blocks. Double-clicking the Blocks icon displays images of each block in the drawing.

Add Content to a Drawing

You can add content from the content area into your current drawing using several methods:

- Drag an item to the graphics area of a drawing to add it using default settings, if any.
- Right-click an item in the content area to display a shortcut menu with several options.
- Double-click a block to display the Insert dialog box; double-click a hatch to display the Boundary Hatch and Fill dialog box.

You can preview graphical content such as a drawing, xref, or block in the content area, and you can display a text description if available.

Update Block Definitions with DesignCenter

Unlike xrefs, when the source file of a block definition is changed, block definitions in the drawings that contain that block are not automatically updated. With DesignCenter, you decide whether a block definition should be updated in the current drawing. The source file of a block definition can be a drawing file or a nested block in a symbol library drawing.

From the shortcut menu displayed when you right-click a block or drawing file in the content area, click Redefine Only or Insert and Redefine to update the selected block.

Open Drawings with DesignCenter

With DesignCenter, you can open a drawing from the content area using the shortcut menu, pressing Ctrl while dragging a drawing, or dragging a drawing icon to any location outside the graphics area of a drawing area. The drawing name is added to the DesignCenter history list for quick access in future sessions.

Add Items from DesignCenter to a Tool Palette

You can add drawings, blocks, and hatches from DesignCenter to the current tool palette.

- From the DesignCenter content area, you can drag one or more items to the current tool palette.
- From the DesignCenter tree view, you can right-click and, from the shortcut menu, create a new tool palette from the current folder, drawing file, or block icon.

When you add drawings to a tool palette, they are inserted as blocks when you drag them into the current drawing.

NOTE You can select multiple blocks or hatches from the content area to add them to a tool palette.

To create a tool palette containing DesignCenter content

1 Click View tab ➤ Palettes panel ➤ DesignCenter. 

2 Do *one* of the following:

- Right-click an item in the DesignCenter tree view. Click Create Tool Palette. The new tool palette contains the drawings, blocks, or hatches from the item you selected.

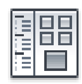
- Right-click the background in the DesignCenter content area. Click Create Tool Palette. The new tool palette contains the drawings, blocks, or hatches from the DesignCenter content area.
- Right-click a drawing in the DesignCenter tree view or content area. Click Create Tool Palette of Blocks. The new tool palette contains the blocks from the drawing you selected.

You can drag additional drawings, blocks, or hatches from the DesignCenter content area to the tool palette.

Command entry: ADCENTER

Toolbar: Standard 

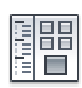
To load the content area from the DesignCenter Search dialog box

1. Click View tab ➤ Palettes panel ➤ DesignCenter. 
2. In DesignCenter, use one of the following methods:
   - Drag the item from the search results list into the content area.
   - Double-click the item in the search results list.
   - Right-click the item in the search results list. Click Load into Content Area.
3. In the DesignCenter content area, double-click the Blocks icon.

Command entry: ADCENTER

Toolbar: Standard 

To load the content area of DesignCenter with a symbol library

1. Click View tab ➤ Palettes panel ➤ DesignCenter. 
2. On the DesignCenter toolbar, click Home.

3 In the content area, double-click the symbol library drawing that you want to load into the DesignCenter and then double-click the Blocks icon.

   The symbol library you selected is loaded into the DesignCenter content area.

NOTE You can set your home folder to any folder that contains symbol library drawings. If your home folder is set to a different path, navigate to a folder that contains symbol library drawings and right-click on the folder. Click Set as Home.

Command entry: ADCENTER

Toolbar: Standard

To load the content area of DesignCenter with hatch patterns

1 Click View tab ➤ Palettes panel ➤ DesignCenter.

2 On the DesignCenter toolbar, click Search.

3 In the Search dialog box, click the Look For box. Click Hatch Pattern Files.

4 On the Hatch Pattern Files tab, in the Search for the Name box, enter *.

5 Click Search Now.

6 Double-click one of the hatch pattern files that was found.

   The hatch pattern file you selected is loaded into DesignCenter.

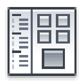
Toolbar: Standard
Command entry: ADCENTER

To open a drawing from DesignCenter

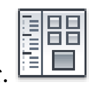
1 Click View tab ➤ Palettes panel ➤ DesignCenter.

2. In DesignCenter, do one of the following:
   - Right-click the drawing icon in the DesignCenter content area. Click Open in Application Window.
   - Press Ctrl and drag the drawing icon from the DesignCenter content area to the drawing area.
   - Drag the drawing icon from the DesignCenter content area to a location anywhere outside the drawing area of the application window. (If you drag the drawing icon into the drawing area, a block is created in the current drawing.)

Command entry: ADCENTER

Toolbar: Standard 

To update a block definition with DesignCenter

1. Click View tab ➤ Palettes panel ➤ DesignCenter. 
2. Right-click a block in the DesignCenter content area. Click Redefine Only or Insert and Redefine.

   NOTE If the source of the block that you want to update is an entire drawing file rather than a block definition within a drawing file, right-click the drawing's icon in the DesignCenter content area. Click Insert as Block.

Command entry: ADCENTER

Toolbar: Standard 

Quick Reference

Commands

ADCENTER

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

ADCNAVIGATE

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

System Variables

ADCSTATE

Indicates whether the DesignCenter window is open or closed.

Utilities

No entries

Command Modifiers

No entries

Retrieve Content from the Web with DesignCenter Online

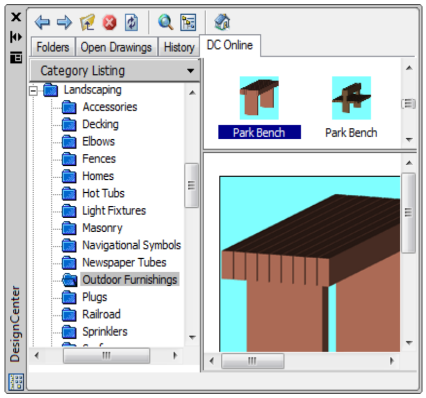
DesignCenter Online provides access to pre-drawn content such as blocks, symbol libraries, manufacturers' content, and online catalogs.

NOTE The DesignCenter Online (DC Online tab) is disabled by default. You can enable it from the CAD Manager Control utility on page 101.

Overview of DesignCenter Online

DesignCenter Online provides access to pre-drawn content such as blocks, symbol libraries, manufacturers' content, and online catalogs. This content can be used in common design applications to assist you in creating your drawings.

To access DesignCenter Online, click the DC Online tab in DesignCenter. Once the DesignCenter Online window is open, you can browse, search, and download content to use in your drawing.

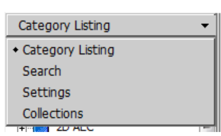


NOTE The DesignCenter Online (DC Online tab) is disabled by default. You can enable it from the CAD Manager Control utility on page 101.

In the DesignCenter Online window, two panes are displayed—a right pane and a left pane. The right pane is called the *content area*. The content area displays the items or folders that you selected in the left pane. The left pane can display one of the following four views:

- **Category Listing.** Displays folders containing libraries of standard parts, manufacturer-specific content, and content aggregator websites.
- **Search.** Searches for online content. You can query items with Boolean and multiple-word search strings.
- **Settings.** Controls how many categories and items are displayed on each page in the content area as a result of a search or folder navigation.
- **Collections.** Specifies the discipline-specific content types that are displayed in DesignCenter Online.

You choose the view by clicking the heading at the top of the left pane.



Once you select a folder in the left pane, all of its content is loaded into the content area. You can select an item in the content area to load it into the preview area. Items can be downloaded by dragging them from the preview area into your drawing or tool palette, or by saving the items to your computer.

NOTE If the DC Online tab is not available in DesignCenter and you want to access DesignCenter Online, see your network or CAD administrator.

DesignCenter Online Privacy

DesignCenter Online is an interactive feature that must be connected to the Internet to deliver content and information. Each time DesignCenter Online is connected, it sends information to Autodesk so that the correct information can be returned. All information is sent anonymously to maintain your privacy.

The following information is sent to Autodesk:

- **Product Name.** The name of the product in which you are using DesignCenter Online
- **Product Release Number.** The version of the product
- **Product Language.** The language version of your product
- **Random Number Identifier.** DesignCenter Online assigns a random number identifier to each person who uses the feature. This identifier is used to retain your Collections and your Settings views each time DesignCenter Online is used.

Autodesk compiles statistics using the information sent from DesignCenter Online to monitor how it is being used and how it can be improved. Autodesk will maintain information provided by or collected from you in accordance with Autodesk's published privacy policy, which is available on *http://www.autodesk.com/privacy*.

Turn the DC Online Tab On or Off

The CAD Manager Control utility turns the DC Online tab in DesignCenter on and off. Information about how to use the utility is available after you install the utility from the Installation Wizard by running the utility and clicking Help in the CAD Manager Control Utility window.

To install the CAD Manager Control utility

1. Insert the product DVD, or the first CD, into your computer's drive.
2. In the Installation Wizard, click Install Tools And Utilities.
3. On the Welcome to the Installation wizard page, click Next.
4. Choose Autodesk CAD Manager Tools 4.0 on the Select the Products to Install page. Click Next.

5 Review the Autodesk software license agreement for your country or region. You must accept this agreement to procede with the installation. Choose your country or region, click I Accept, and then click Next.

NOTE If you do not agree to the terms of the license and wish to terminate the installation, click Cancel.

6 On the Review - Configure - Install page, click Install if you want to accept the default install location.

If you want the utility installed in a different installation path, click Configure, set the path and then click Configuration Complete and then Install.

7 Click Finish when the Installation Complete page is displayed.

See also:

- Use Autodesk Seek to Add and Share Drawings

Quick Reference

Commands

ADCENTER

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

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Understand DesignCenter Online Content Types

With DesignCenter Online, content is categorized into folders.

In the DesignCenter Online folders, you can retrieve discipline-specific content. The content that you can retrieve includes the following:

- **Standard Parts.** Generic standard parts that are commonly used in design. These parts include blocks for architectural, mechanical, and GIS applications.
- **Manufacturers.** Blocks and 3D models that can be located and downloaded by clicking a link to a manufacturer's website.
- **Aggregators**. Lists of libraries from commercial catalog providers can be searched for parts and blocks.

You use the Collections view to select the categories of online content that you want to display in the Category Listing view.

NOTE The DesignCenter Online (DC Online tab) is disabled by default. You can enable it from the CAD Manager Control utility on page 101.

To view online content folders in the Category Listing view

- In DesignCenter Online, at the top of the left pane, click the heading, and then click Category Listing.

The category folders are displayed in the left pane of the window.

Quick Reference

Commands

No entries

System Variables

No entries

Utilities

No entries

Command Modifiers

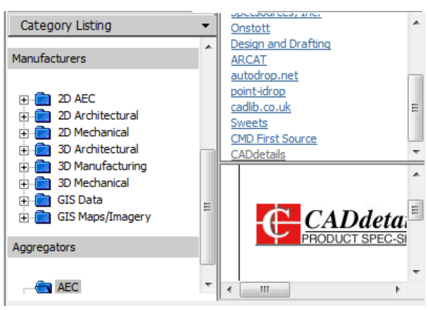
No entries

Retrieve Content from the Web

You can download content from the Web and use it in your drawings.

Browse for Content

When you use the Category Listing view, you can click the folders in the left pane to view their contents. These folders may contain other folders.



When you click a folder or an item inside a folder, the contents are displayed in the Content area. When you click a block, graphical and descriptive information about the block is displayed in the preview area.

Search for Content

When you search for online content with DesignCenter Online, you can query items with Boolean and multiple-word search strings in the Search view. You can access Search by clicking the magnifying glass or by choosing Search from the drop-down heading at the top of the left pane.

Control the Number of Categories and Items in a Page

By using the Settings view, you can control how many categories or items are displayed on each page in the content area as a result of a search or folder navigation.

Collections

You can choose the type of content to navigate and search. In the Collections pane, you can specify the content types that are displayed each time you open DesignCenter Online. For example, if you use architectural blocks in your drawings, you select collections that contain architectural items. Once you make your selection, categories that you specified are displayed.

Download Content

To download content from the Web, locate the folder containing the content that you want to use. Then, click a thumbnail image of the content in the content area. The content is displayed in the preview area along with information about the content. You can drag the block directly from the preview area into a drawing or tool palette, or you can save it to your computer to be used later.

See also:

- Use Autodesk Seek to Add and Share Drawings

To search for content in DesignCenter Online

1. In DesignCenter Online, at the top of the left pane, click the heading, and then click Search.
2. In the Search view, enter a single word or multiple-word strings.

NOTE The Need Help link provides more information about searches, including examples of Boolean searches.

To specify content collections

1. In DesignCenter Online, at the top of the left pane, click the heading, and then click Collections.
2. In the Collections view, click the check boxes of the collections that you want to use.
3. Click Update Collections.

   The categories that you have selected are displayed in the left pane.

To download content to your computer

1. In DesignCenter Online, at the top of the left pane, click the heading, and then click Category Listing.
2. In the Category folders, click a content item.
3. In the Preview area, under the image of the content item, click Save This Symbol As.
4. In the Save As dialog box, specify the location on your computer and the file name.
5. Click Save.

   The content is downloaded to your computer.

To download content to your drawing

1. In DesignCenter Online, at the top of the left pane, click the heading, and then click Category Listing.
2. In the Category folders, click a content item to display it in the Preview area.
3. Drag the image from the Preview area into your drawing or tool palette.

Quick Reference

Commands

ADCENTER

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

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Customize the Drawing Environment

4

You can change many window and drawing environment settings in the Options dialog box. For example, you can change how often a drawing is automatically saved to a temporary file, and you can link the program to folders containing files you use frequently. You can create workspaces to set up a drawing environment that is specific to your drawing needs. Experiment with different settings until you create the drawing environment that best fits your needs.

Set Interface Options

You can adjust the application interface and drawing area to match the way you work.

Set Up the Drawing Area

You can adjust the color and display schemes used in the application and drawing windows, and control the behavior of general features such as zoom transitions.

Many of the settings are available from shortcut menus and the Options dialog box. Some workspace elements, such as the presence and location of toolbars and palettes, can be specified and saved using the Customize User Interface dialog box.

Some settings affect how you work in the drawing area:

- **Background Colors (Options dialog box, Display tab)**. You specify the background colors used in the layout and Model tabs and the color used for prompts and crosshairs.

- **Color Scheme (Options dialog box, Display tab, Colors)**. You specify a dark or light color scheme for the overall user interface. The settings affect the window frame background, status bar, title bar, menu browser frame, toolbars, and palettes.
- **Background Colors (Options dialog box, Display tab, Colors)**. You specify the background colors used in model space, layouts, and the block editor. Background colors on the Model tab change to indicate whether you are working in a 2D design context, 3D modeling (parallel projection), or 3D modeling (perspective projection).
- **UCS Icon and Crosshairs Cursor (Options dialog box, 3D Modeling tab).** You specify that the 3D display options and labels for the UCS icon can be set in the 3D Modeling tab of the Options dialog box.
- **Color Assignments for X, Y, and Z (Options dialog box, Display tab, Colors)**. In 3D views, any interface elements that are associated with the UCS *X*, *Y*, and *Z* axis use special color assignments. The *X* axis is colored or tinted red, the *Y* axis is green, and the *Z* axis is blue. These tints can be turned on or off in the Drawing Window Colors dialog box.
- **Clean Screen.** You can expand the drawing display area to display only the menu bar, status bar, and command window with the clean screen button on the application status bar. Click the button again to restore the previous setup.
- **View Transitions.** You can control whether view transitions are smooth or instantaneous when you pan, zoom, or change from one view to another (VTOPTIONS command). The default is a smooth transition.

Tooltips

Several types of tooltips provide pop-up information for interaction with toolbars, object snaps, and drafting operations.

You can view tooltips in toolbars, the menu browser, the ribbon, and dialog boxes. Initially, a basic tooltip is displayed. If you continue to hover, the tooltip expands to display additional information. You can customize the display and content of a tooltip.

See also:

- Display Tab (Options Dialog Box) in the *Command Reference*
- User Interface Customization in the *Customization Guide*

- Rollover Tooltips in the *Customization Guide*
- Create Tooltips and Extended Help for Commands in the *Customization Guide*

To set options

1. Click the Application button. At the bottom of the Application menu, click Options.
2. In the Options dialog box, click a tab.
3. Set options as desired.
4. Do either or both of the following:
   - Click Apply to record the current options settings in the system registry.
   - Click OK to record the current options settings in the system registry and close the Options dialog box.

Command entry: OPTIONS

To customize the colors of the application window elements

1. Click the Application button. At the bottom of the Application menu, click Options.
2. In the Options dialog box, Display tab, click Colors.
3. In the Drawing Window Colors dialog box, select context and then the interface element you want to change.
4. Select the color you want to use from the Color list.

   To specify a custom color, select Select Color from the Color list.
5. If you want to revert to the default colors, click Restore Current Element, Restore Current Context, or Restore All Contexts.
6. Click Apply and Close to record the current option settings in the system registry and close the dialog box.
7. Click OK to close the Options dialog box.

Command entry: OPTIONS

To change the appearance of view transitions

1. At the Command prompt, enter **vtoptions**.
2. In the View Transitions dialog box, check one or more of the following options:
   - **Enable Animation for Pan and Zoom**. Makes a smooth view transition during panning and zooming.
   - **Enable Animation When View Rotates**. Makes a smooth view transition when the view angle is changed.
   - **Enable Animation During Scripts**. Makes a smooth view transition while a script is running.
3. Set the transition speed by moving the slider.
4. To preserve performance, set the minimum frames per second for showing smooth view transitions. When a smooth view transition cannot maintain this speed, an instant transition is used.
5. Click OK.

Command entry: VTOPTIONS

To display hidden message dialogs

1. Click the Application button. At the bottom of the Application menu, click Options.
2. In the Options dialog box, System tab, under General Options, click Hidden Messages Settings button.
3. The Hidden Message Settings dialog box is displayed.
4. Check the corresponding dialog name from the tree directory.
5. Click OK.

To turn tooltips on or off

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Display tab, under Window Elements, click Show Tooltips.
3. Click OK.

Quick Reference

Commands

3DCONFIG

Sets options that affect 3D display performance.

CLEANSCREENON

Clears the screen of toolbars and dockable windows (excluding the command line).

CLEANSCREENOFF

Restores display of toolbars and dockable windows (excluding the command line).

OPTIONS

Customizes the program settings.

VIEWRES

Sets the resolution for objects in the current viewport.

VTOPTIONS

Displays a change in view as a smooth transition.

System Variables

CALCINPUT

Controls whether mathematical expressions and global constants are evaluated in text and numeric entry boxes of windows and dialog boxes.

CLEANSCREENSTATE

Indicates whether the clean screen state is on or off.

CURSORSIZE

Determines the size of the crosshairs as a percentage of the screen size.

DRAGMODE

Controls the way dragged objects are displayed.

EXTNAMES

Sets the parameters for named object names (such as linetypes and layers) stored in definition tables.

GRIPCOLOR

Controls the color of unselected grips.

GRIPHOT

Controls the color of selected grips.

GRIPS

Controls the use of selection set grips for the Stretch, Move, Rotate, Scale, and Mirror Grip modes.

INSUNITS

Specifies a drawing-units value for automatic scaling of blocks, images, or xrefs when inserted or attached to a drawing.

INSUNITSDEFSOURCE

Sets source content units value when INSUNITS is set to 0.

INSUNITSDEFTARGET

Sets target drawing units value when INSUNITS is set to 0.

INTELLIGENTUPDATE

Controls the graphics refresh rate.

ISAVEBAK

Improves the speed of incremental saves, especially for large drawings.

ISAVEPERCENT

Determines the amount of wasted space tolerated in a drawing file.

LAYOUTREGENCTL

Specifies how the display list is updated in the Model tab and layout tabs.

LOCALE

Displays a code that indicates the current locale.

LOCALROOTPREFIX

Stores the full path to the root folder where local customizable files were installed.

LOCKUI

Locks the position and size of toolbars and dockable windows such as DesignCenter and the Properties palette.

LOGFILEMODE

Specifies whether the contents of the text window are written to a log file.

LOGFILENAME

Specifies the path and name of the text window log file for the current drawing.

LOGFILEPATH

Specifies the path for the text window log files for all drawings in a session.

MTEXTED

Sets the application for editing multiline text objects.

OLEQUALITY

Sets the default plot quality for OLE objects.

OLESTARTUP

Controls whether the source application of an embedded OLE object loads when plotting.

OSNAPCOORD

Controls whether coordinates entered on the command line will override running object snaps.

PAPERUPDATE

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.

PALETTEOPAQUE

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.

PICKADD

Controls whether subsequent selections replace the current selection set or add to it.

PICKAUTO

Controls automatic windowing at the Select Objects prompt.

PICKBOX

Sets the object selection target height, in pixels.

PICKDRAG

Controls the method of drawing a selection window.

PICKFIRST

Controls whether you select objects before (noun-verb selection) or after you issue a command.

PICKSTYLE

Controls the use of group selection and associative hatch selection.

PSTYLEPOLICY

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.

QTEXTMODE

Controls how text is displayed.

RASTERPREVIEW

Controls whether BMP preview images are saved with the drawing.

ROLLOVERTIPS

Controls the display of rollover tooltips in the application.

SAVEFILE

Stores the current automatic save file name.

SAVEFILEPATH

Specifies the path to the directory for all automatic save files for the current session.

SAVETIME

Sets the automatic save interval, in minutes.

SPLINESEGS

Sets the number of line segments to be generated for each spline-fit polyline generated by the Spline option of the PEDIT command.

TDUSRTIMER

Stores the user-elapsed timer.

TOOLTIPMERGE

Combines drafting tooltips into a single tooltip.

TOOLTIPS

Controls the display of tooltips on the ribbon, toolbars, and other user interface elements.

VISRETAIN

Controls the properties of xref-dependent layers.

VTDURATION

Sets the duration of a smooth view transition, in milliseconds.

VTENABLE

Controls when smooth view transitions are used.

VTFPS

Sets the minimum speed of a smooth view transition, in frames per second.

XLOADCTL

Turns xref demand-loading on and off, and controls whether it opens the referenced drawing or a copy.

Utilities

No entries

Command Modifiers

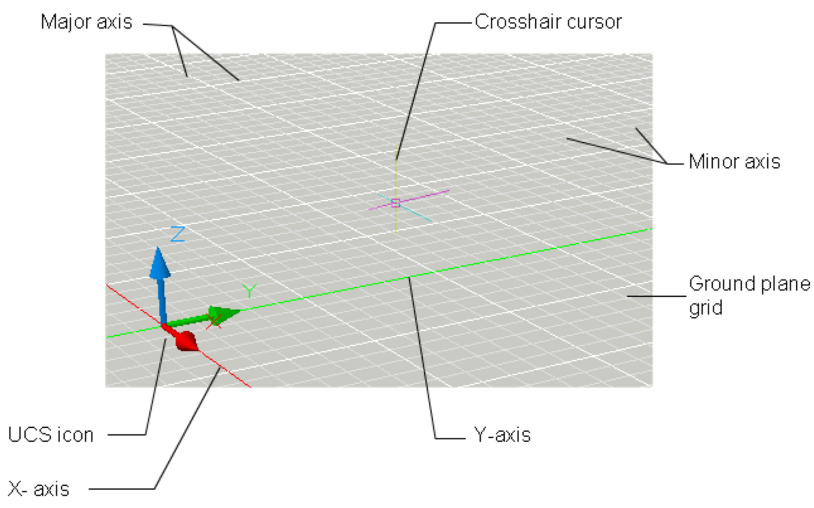
No entries

Set Options for 3D Modeling with Perspective Projection

You can set specific display options when working with 3D models using perspective projection.

Using the Options dialog box, you can specify the following options when your 3D model is set to use a perspective view:

- **Ground Plane (Options dialog box, Display tab, Colors).** When perspective projection is turned on, the XY plane of the UCS displays as a *ground plane* with a gradient color. The ground plane displays a gradient from the *ground horizon* to the *ground origin*.
- **Sky (Options dialog box, Display tab, Colors).** The area not covered by the ground plane is the *sky*, which displays a gradient color from the *sky horizon* to the *sky zenith*.
- **Underground (Options dialog box, Display tab, Colors).** If the ground plane is viewed from below ground, the ground plane displays a gradient from the *earth horizon* to the *earth azimuth*.
- **Ground Plane Grid (Options dialog box, Display tab, Colors).** When perspective projection is turned on, the grid displays as a *ground plane grid*. Colors are set for major grid lines, minor grid lines, and axis lines.



Quick Reference

Commands

3DCONFIG

Sets options that affect 3D display performance.

OPTIONS

Customizes the program settings.

VIEWRES

Sets the resolution for objects in the current viewport.

System Variables

APERTURE

Sets the display size for the object snap target box, in pixels.

CALCINPUT

Controls whether mathematical expressions and global constants are evaluated in text and numeric entry boxes of windows and dialog boxes.

CLEANSCREENSTATE

Indicates whether the clean screen state is on or off.

CURSORSIZE

Determines the size of the crosshairs as a percentage of the screen size.

DCTCUST

Displays the path and file name of the current custom spelling dictionary.

DCTMAIN

Displays the three letter keyword for the current main spelling dictionary.

DEFPLSTYLE

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.

DRAGMODE

Controls the way dragged objects are displayed.

EXTNAMES

Sets the parameters for named object names (such as linetypes and layers) stored in definition tables.

FILLMODE

Specifies whether hatches and fills, 2D solids, and wide polylines are filled in.

GRIPBLOCK

Controls the display of grips in blocks.

GRIPCOLOR

Controls the color of unselected grips.

GRIPHOT

Controls the color of selected grips.

GRIPS

Controls the use of selection set grips for the Stretch, Move, Rotate, Scale, and Mirror Grip modes.

INSUNITS

Specifies a drawing-units value for automatic scaling of blocks, images, or xrefs when inserted or attached to a drawing.

INSUNITSDEFSOURCE

Sets source content units value when INSUNITS is set to 0.

INSUNITSDEFTARGET

Sets target drawing units value when INSUNITS is set to 0.

INTELLIGENTUPDATE

Controls the graphics refresh rate.

ISAVEBAK

Improves the speed of incremental saves, especially for large drawings.

ISAVEPERCENT

Determines the amount of wasted space tolerated in a drawing file.

LAYOUTREGENCTL

Specifies how the display list is updated in the Model tab and layout tabs.

LOCALE

Displays a code that indicates the current locale.

LOCALROOTPREFIX

Stores the full path to the root folder where local customizable files were installed.

LOCKUI

Locks the position and size of toolbars and dockable windows such as DesignCenter and the Properties palette.

LOGFILEMODE

Specifies whether the contents of the text window are written to a log file.

LOGFILENAME

Specifies the path and name of the text window log file for the current drawing.

LOGFILEPATH

Specifies the path for the text window log files for all drawings in a session.

MTEXTED

Sets the application for editing multiline text objects.

OLEQUALITY

Sets the default plot quality for OLE objects.

OLESTARTUP

Controls whether the source application of an embedded OLE object loads when plotting.

OSNAPCOORD

Controls whether coordinates entered on the command line will override running object snaps.

PAPERUPDATE

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.

PALETTEOPAQUE

Controls whether palettes can be made transparent.

PICKADD

Controls whether subsequent selections replace the current selection set or add to it.

PICKAUTO

Controls automatic windowing at the Select Objects prompt.

PICKBOX

Sets the object selection target height, in pixels.

PICKDRAG

Controls the method of drawing a selection window.

PICKFIRST

Controls whether you select objects before (noun-verb selection) or after you issue a command.

PICKSTYLE

Controls the use of group selection and associative hatch selection.

PSTYLEPOLICY

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.

QTEXTMODE

Controls how text is displayed.

RASTERPREVIEW

Controls whether BMP preview images are saved with the drawing.

ROLLOVERTIPS

Controls the display of rollover tooltips in the application.

SAVEFILEPATH

Stores the current automatic save file name.

SAVEFILEPATH

Specifies the path to the directory for all automatic save files for the current session.

SAVETIME

Sets the automatic save interval, in minutes.

SPLINESEGS

Sets the number of line segments to be generated for each spline-fit polyline generated by the Spline option of the PEDIT command.

TDUSRTIMER

Stores the user-elapsed timer.

TOOLTIPMERGE

Combines drafting tooltips into a single tooltip.

TOOLTIPS

Controls the display of tooltips on the ribbon, toolbars, and other user interface elements.

VISRETAIN

Controls the properties of xref-dependent layers.

VTDURATION

Sets the duration of a smooth view transition, in milliseconds.

VTENABLE

Controls when smooth view transitions are used.

VTFPS

Sets the minimum speed of a smooth view transition, in frames per second.

XLOADCTL

Turns xref demand-loading on and off, and controls whether it opens the referenced drawing or a copy.

Utilities

No entries

Command Modifiers

No entries

Switch Between Model Space and Layouts

You can control how you change between model space and one or more layouts.

The classic interface provides a Model tab and one or more layout tabs. To optimize space in the drawing area, you can turn off these tabs and use the equivalent buttons on the status bar. The control to change between the two interface designs is included as an item on the Model and layout tab shortcut menu, and on the shortcut menu of the Model/Layout button on the status bar.

NOTE Access to all shortcut menu options is available from the tabs only.

To change the Model and layout tabs to status bar buttons

- Right-click the Model tab or a layout tab. Click Hide Layout and Model Tabs.

To turn on the Model and layout tabs

- On the status bar, right-click the Model or layout button. Click Display Layout and Model Tabs.

Quick Reference

Commands

DRAGMODE

Controls the way dragged objects are displayed.

OPTIONS

Customizes the program settings.

VIEWRES

Sets the resolution for objects in the current viewport.

System Variables

APERTURE

Sets the display size for the object snap target box, in pixels.

CALCINPUT

Controls whether mathematical expressions and global constants are evaluated in text and numeric entry boxes of windows and dialog boxes.

CLEANSCREENSTATE

Indicates whether the clean screen state is on or off.

CURSORSIZE

Determines the size of the crosshairs as a percentage of the screen size.

DCTCUST

Displays the path and file name of the current custom spelling dictionary.

DCTMAIN

Displays the three letter keyword for the current main spelling dictionary.

DEFPLSTYLE

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.

DRAGMODE

Controls the way dragged objects are displayed.

EXTNAMES

Sets the parameters for named object names (such as linetypes and layers) stored in definition tables.

FILLMODE

Specifies whether hatches and fills, 2D solids, and wide polylines are filled in.

GRIPBLOCK

Controls the display of grips in blocks.

GRIPCOLOR

Controls the color of unselected grips.

GRIPHOT

Controls the color of selected grips.

GRIPS

Controls the use of selection set grips for the Stretch, Move, Rotate, Scale, and Mirror Grip modes.

INSUNITS

Specifies a drawing-units value for automatic scaling of blocks, images, or xrefs when inserted or attached to a drawing.

INSUNITSDEFSOURCE

Sets source content units value when INSUNITS is set to 0.

INSUNITSDEFTARGET

Sets target drawing units value when INSUNITS is set to 0.

INTELLIGENTUPDATE

Controls the graphics refresh rate.

ISAVEBAK

Improves the speed of incremental saves, especially for large drawings.

ISAVEPERCENT

Determines the amount of wasted space tolerated in a drawing file.

LAYOUTREGENCTL

Specifies how the display list is updated in the Model tab and layout tabs.

LOCALE

Displays a code that indicates the current locale.

LOCALROOTPREFIX

Stores the full path to the root folder where local customizable files were installed.

LOCKUI

Locks the position and size of toolbars and dockable windows such as DesignCenter and the Properties palette.

LOGFILEMODE

Specifies whether the contents of the text window are written to a log file.

LOGFILENAME

Specifies the path and name of the text window log file for the current drawing.

LOGFILEPATH

Specifies the path for the text window log files for all drawings in a session.

MTEXTED

Sets the application for editing multiline text objects.

OLEQUALITY

Sets the default plot quality for OLE objects.

OLESTARTUP

Controls whether the source application of an embedded OLE object loads when plotting.

OSNAPCOORD

Controls whether coordinates entered on the command line will override running object snaps.

PAPERUPDATE

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.

PALETTEOPAQUE

Controls whether palettes can be made transparent.

PICKADD

Controls whether subsequent selections replace the current selection set or add to it.

PICKAUTO

Controls automatic windowing at the Select Objects prompt.

PICKBOX

Sets the object selection target height, in pixels.

PICKDRAG

Controls the method of drawing a selection window.

PICKFIRST

Controls whether you select objects before (noun-verb selection) or after you issue a command.

PICKSTYLE

Controls the use of group selection and associative hatch selection.

PSTYLEPOLICY

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.

QTEXTMODE

Controls how text is displayed.

RASTERPREVIEW

Controls whether BMP preview images are saved with the drawing.

ROLLOVERTIPS

Controls the display of rollover tooltips in the application.

SAVEFILEPATH

Stores the current automatic save file name.

SAVEFILEPATH

Specifies the path to the directory for all automatic save files for the current session.

SAVETIME

Sets the automatic save interval, in minutes.

SPLINESEGS

Sets the number of line segments to be generated for each spline-fit polyline generated by the Spline option of the PEDIT command.

TDUSRTIMER

Stores the user-elapsed timer.

TOOLTIPMERGE

Combines drafting tooltips into a single tooltip.

TOOLTIPS

Controls the display of tooltips on the ribbon, toolbars, and other user interface elements.

VISRETAIN

Controls the properties of xref-dependent layers.

VTDURATION

Sets the duration of a smooth view transition, in milliseconds.

VTENABLE

Controls when smooth view transitions are used.

VTFPS

Sets the minimum speed of a smooth view transition, in frames per second.

XLOADCTL

Turns xref demand-loading on and off, and controls whether it opens the referenced drawing or a copy.

Utilities

No entries

Command Modifiers

No entries

Specify Application Fonts

Change the fonts used in the application window and in the text window.

You can specify the font that is displayed in both the application and text windows. To change the application font, use the Options dialog box, Display tab.

NOTE This setting does not affect the text in your drawings.

To change the font displayed in the Command window

1. Click the Application button. At the bottom of the Application menu, click Options.
2. In the Options dialog box, Display tab, under Window Elements, click Fonts.
3. In the Command Line Window Font dialog box, select the appropriate Font, Font Style, and Size.

   An example of the current choices appears under Sample Command Line Font.
4. Click Apply & Close to record the current option settings in the system registry and close the dialog box.
5. In the Options dialog box, click OK.

Command entry: OPTIONS

Quick Reference

Commands

OPTIONS

Customizes the program settings.

System Variables

APERTURE

Sets the display size for the object snap target box, in pixels.

CALCINPUT

Controls whether mathematical expressions and global constants are evaluated in text and numeric entry boxes of windows and dialog boxes.

CLEANSCREENSTATE

Indicates whether the clean screen state is on or off.

CURSORSIZE

Determines the size of the crosshairs as a percentage of the screen size.

DCTCUST

Displays the path and file name of the current custom spelling dictionary.

DCTMAIN

Displays the three letter keyword for the current main spelling dictionary.

DEFPLSTYLE

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.

DRAGMODE

Controls the way dragged objects are displayed.

EXTNAMES

Sets the parameters for named object names (such as linetypes and layers) stored in definition tables.

FILLMODE

Specifies whether hatches and fills, 2D solids, and wide polylines are filled in.

GRIPBLOCK

Controls the display of grips in blocks.

GRIPCOLOR

Controls the color of unselected grips.

GRIPHOT

Controls the color of selected grips.

GRIPS

Controls the use of selection set grips for the Stretch, Move, Rotate, Scale, and Mirror Grip modes.

INSUNITS

Specifies a drawing-units value for automatic scaling of blocks, images, or xrefs when inserted or attached to a drawing.

INSUNITSDEFSOURCE

Sets source content units value when INSUNITS is set to 0.

INSUNITSDEFTARGET

Sets target drawing units value when INSUNITS is set to 0.

INTELLIGENTUPDATE

Controls the graphics refresh rate.

ISAVEBAK

Improves the speed of incremental saves, especially for large drawings.

ISAVEPERCENT

Determines the amount of wasted space tolerated in a drawing file.

LAYOUTREGENCTL

Specifies how the display list is updated in the Model tab and layout tabs.

LOCALE

Displays a code that indicates the current locale.

LOCALROOTPREFIX

Stores the full path to the root folder where local customizable files were installed.

LOCKUI

Locks the position and size of toolbars and dockable windows such as DesignCenter and the Properties palette.

LOGFILEMODE

Specifies whether the contents of the text window are written to a log file.

LOGFILENAME

Specifies the path and name of the text window log file for the current drawing.

LOGFILEPATH

Specifies the path for the text window log files for all drawings in a session.

MTEXTED

Sets the application for editing multiline text objects.

OLEQUALITY

Sets the default plot quality for OLE objects.

OLESTARTUP

Controls whether the source application of an embedded OLE object loads when plotting.

OSNAPCOORD

Controls whether coordinates entered on the command line will override running object snaps.

PAPERUPDATE

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.

PALETTEOPAQUE

Controls whether palettes can be made transparent.

PICKADD

Controls whether subsequent selections replace the current selection set or add to it.

PICKAUTO

Controls automatic windowing at the Select Objects prompt.

PICKBOX

Sets the object selection target height, in pixels.

PICKDRAG

Controls the method of drawing a selection window.

PICKFIRST

Controls whether you select objects before (noun-verb selection) or after you issue a command.

PICKSTYLE

Controls the use of group selection and associative hatch selection.

PSTYLEPOLICY

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.

QTEXTMODE

Controls how text is displayed.

RASTERPREVIEW

Controls whether BMP preview images are saved with the drawing.

ROLLOVERTIPS

Controls the display of rollover tooltips in the application.

SAVEFILEPATH

Stores the current automatic save file name.

SAVEFILEPATH

Specifies the path to the directory for all automatic save files for the current session.

SAVETIME

Sets the automatic save interval, in minutes.

SPLINESEGS

Sets the number of line segments to be generated for each spline-fit polyline generated by the Spline option of the PEDIT command.

TDUSRTIMER

Stores the user-elapsed timer.

TOOLTIPMERGE

Combines drafting tooltips into a single tooltip.

TOOLTIPS

Controls the display of tooltips on the ribbon, toolbars, and other user interface elements.

VISRETAIN

Controls the properties of xref-dependent layers.

VTDURATION

Sets the duration of a smooth view transition, in milliseconds.

VTENABLE

Controls when smooth view transitions are used.

VTFPS

Sets the minimum speed of a smooth view transition, in frames per second.

XLOADCTL

Turns xref demand-loading on and off, and controls whether it opens the referenced drawing or a copy.

Utilities

No entries

Command Modifiers

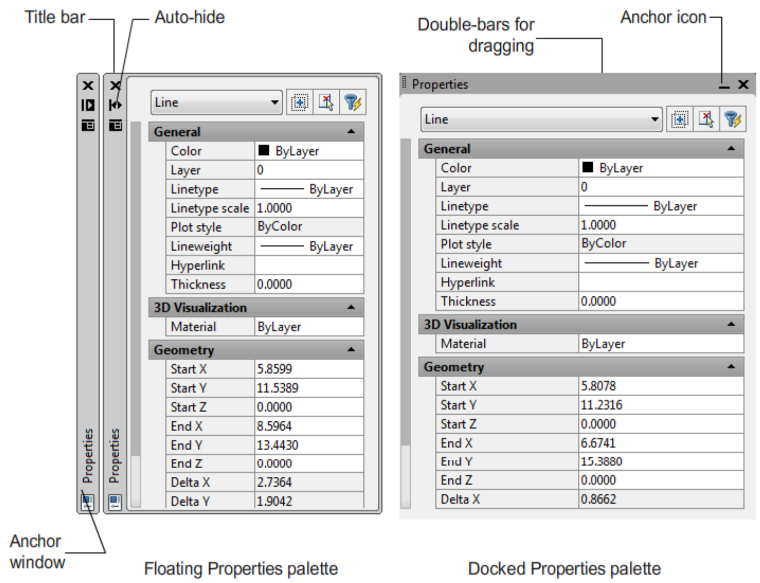
No entries

Specify the Behavior of Dockable Windows

Windows such as the Properties palette, tool palettes, and DesignCenter can be docked, anchored, or floated.

Settings for these and other options are often changed on a shortcut menu, available by right-clicking the title bar of the palette or window.

- **Resize**. Drag an edge of the window to change its size. If the window has panes, drag the bar between panes to resize the panes.
- **Allow Docking**. Select this option if you want to dock or anchor a dockable window. A docked window adheres to one side of the application window, causing the drawing area to be resized.
- **Anchor.** Attach, or anchor, a dockable window or palette to the left or right side of the drawing area. An anchored window rolls open and closed as the cursor moves across it. When an anchored window is open, its content overlaps the drawing area. An anchored window cannot be set to stay open. The Allow Docking option must be selected before you can anchor a window.
- **Auto-hide**. A floating window rolls open and closes as the cursor moves across it. When this option is cleared, the window stays open continuously. Docked windows with auto-hide show up as a bar inside the application.
- **Transparency**. Sets the degree of transparency for the window and on mouse over. The window becomes transparent so that it does not obscure objects under it. The window becomes more opaque when it is moused over. This option is not available for all windows.



Floating Properties palette

Docked Properties palette

You can hide all the palettes at once with HIDEPALETTES and turn on all hidden palettes with SHOWPALETTES.

NOTE If a palette has been turned back on manually and moved, it is not affected by SHOWPALETTES.

To anchor a dockable window

1. At the top of the window or palette title bar, click the Propterties button. Click Allow Docking.
2. Click the Properties button again. Click Anchor Right or Anchor Left.

To float an anchored window

Do one of the following:

- At the top of the window or palette title bar, click the Properties button. Clear Allow Docking.
- When the anchored window is open, drag the window title bar away from the anchor tab base.
- Double-click the anchor tab.

To dock a window or palette

1 At the top of the window or palette title bar, click the Properties button. Click Allow Docking.

2 Click and drag the window or palette to a docking location on the right or left side of the drawing area.

3 When the outline of the window is displayed in the docking area, release the button.

NOTE To place a toolbar in a docking region without docking it, hold down the Ctrl key as you drag.

To undock a window or palette

Use one of the following methods:

- At the top of the window or palette title bar, click the Properties button. Clear Allow Docking.
- Double-click the double bars on the side or top of the window.
- Position the cursor on the double bars at the top or side of the window, hold down the left button on your pointing device, and drag the window away from its docked location.

To turn Auto-hide on or off for a floating palette or window

- At the top of the window or palette title bar, click the Auto-hide button.

If Auto-hide is selected, floating windows roll open and closed as the cursor moves across them. When this option is cleared, the full window stays open continuously.

NOTE This procedure applies to the Tool palette, DesignCenter, the Properties palette, and several other palettes.

Quick Reference

Commands

DRAGMODE

Controls the way dragged objects are displayed.

HIDEPALETTES

Hides currently displayed palettes (including the command line).

LOGFILEOFF

Closes the text window log file opened by LOGFILEON.

LOGFILEON

Writes the text window contents to a file.

OPTIONS

Customizes the program settings.

SHOWPALETTES

Restores the display of hidden palettes.

VIEWRES

Sets the resolution for objects in the current viewport.

VTOPTIONS

Displays a change in view as a smooth transition.

System Variables

APERTURE

Sets the display size for the object snap target box, in pixels.

CALCINPUT

Controls whether mathematical expressions and global constants are evaluated in text and numeric entry boxes of windows and dialog boxes.

CLEANSCREENSTATE

Indicates whether the clean screen state is on or off.

CURSORSIZE

Determines the size of the crosshairs as a percentage of the screen size.

DCTCUST

Displays the path and file name of the current custom spelling dictionary.

DCTMAIN

Displays the three letter keyword for the current main spelling dictionary.

DEFPLSTYLE

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.

DRAGMODE

Controls the way dragged objects are displayed.

EXTNAMES

Sets the parameters for named object names (such as linetypes and layers) stored in definition tables.

FILLMODE

Specifies whether hatches and fills, 2D solids, and wide polylines are filled in.

GRIPBLOCK

Controls the display of grips in blocks.

GRIPCOLOR

Controls the color of unselected grips.

GRIPHOT

Controls the color of selected grips.

GRIPS

Controls the use of selection set grips for the Stretch, Move, Rotate, Scale, and Mirror Grip modes.

INSUNITS

Specifies a drawing-units value for automatic scaling of blocks, images, or xrefs when inserted or attached to a drawing.

INSUNITSDEFSOURCE

Sets source content units value when INSUNITS is set to 0.

INSUNITSDEFTARGET

Sets target drawing units value when INSUNITS is set to 0.

INTELLIGENTUPDATE

Controls the graphics refresh rate.

ISAVEBAK

Improves the speed of incremental saves, especially for large drawings.

ISAVEPERCENT

Determines the amount of wasted space tolerated in a drawing file.

LAYOUTREGENCTL

Specifies how the display list is updated in the Model tab and layout tabs.

LOCALE

Displays a code that indicates the current locale.

LOCALROOTPREFIX

Stores the full path to the root folder where local customizable files were installed.

LOCKUI

Locks the position and size of toolbars and dockable windows such as DesignCenter and the Properties palette.

LOGFILEMODE

Specifies whether the contents of the text window are written to a log file.

LOGFILENAME

Specifies the path and name of the text window log file for the current drawing.

LOGFILEPATH

Specifies the path for the text window log files for all drawings in a session.

MTEXTED

Sets the application for editing multiline text objects.

OLEQUALITY

Sets the default plot quality for OLE objects.

OLESTARTUP

Controls whether the source application of an embedded OLE object loads when plotting.

OSNAPCOORD

Controls whether coordinates entered on the command line will override running object snaps.

PAPERUPDATE

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.

PALETTEOPAQUE

Controls whether palettes can be made transparent.

PICKADD

Controls whether subsequent selections replace the current selection set or add to it.

PICKAUTO

Controls automatic windowing at the Select Objects prompt.

PICKBOX

Sets the object selection target height, in pixels.

PICKDRAG

Controls the method of drawing a selection window.

PICKFIRST

Controls whether you select objects before (noun-verb selection) or after you issue a command.

PICKSTYLE

Controls the use of group selection and associative hatch selection.

PSTYLEPOLICY

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.

QTEXTMODE

Controls how text is displayed.

RASTERPREVIEW

Controls whether BMP preview images are saved with the drawing.

ROLLOVERTIPS

Controls the display of rollover tooltips in the application.

SAVEFILEPATH

Stores the current automatic save file name.

SAVEFILEPATH

Specifies the path to the directory for all automatic save files for the current session.

SAVETIME

Sets the automatic save interval, in minutes.

SPLINESEGS

Sets the number of line segments to be generated for each spline-fit polyline generated by the Spline option of the PEDIT command.

TDUSRTIMER

Stores the user-elapsed timer.

TOOLTIPMERGE

Combines drafting tooltips into a single tooltip.

TOOLTIPS

Controls the display of tooltips on the ribbon, toolbars, and other user interface elements.

VISRETAIN

Controls the properties of xref-dependent layers.

VTDURATION

Sets the duration of a smooth view transition, in milliseconds.

VTENABLE

Controls when smooth view transitions are used.

VTFPS

Sets the minimum speed of a smooth view transition, in frames per second.

XLOADCTL

Turns xref demand-loading on and off, and controls whether it opens the referenced drawing or a copy.

Utilities

No entries

Command Modifiers

No entries

Control the Display of Toolbars

Similar to dockable windows, you can control the behavior of toolbars.

To display or hide toolbars, right-click any toolbar to display a list of toolbars. A check mark next to a toolbar name indicates that it is displayed. Click a toolbar name in the list to display or clear the check mark.

A toolbar can be docked or floating. A docked toolbar is attached to any edge of the drawing area. A toolbar docked at the top edge of the drawing area is located below the ribbon. Undock a toolbar by clicking the double bars and dragging it into the drawing area. You can click the title bar and drag it to a new location or dock it. Resize a floating toolbar by dragging an edge.

Lock the Position of Toolbars and Dockable Windows

Once you have arranged toolbars and docked, floating, or anchored windows the way you want them, you can lock their position. Locked toolbars and windows can still be opened and closed and items can be added and deleted. To unlock them temporarily, press and hold *Ctrl*.

To lock the position and size of toolbars and dockable windows

- Do one of the following:
  - Click View tab ➤ Windows panel ➤ Window Locking ➤ Floating Toolbars/Panels.
  - Click View tab ➤ Windows panel ➤ Window Locking ➤ Docked Toolbars/Panels.
  - Click View tab ➤ Windows panel ➤ Window Locking ➤ Floating Windows.
  - Click View tab ➤ Windows panel ➤ Window Locking ➤ Docked Windows.

A lock icon in the system tray indicates whether toolbars or dockable windows are locked. To unlock them temporarily, hold down Ctrl.

Command entry: LOCKUI
Shortcut menu: Right-click the lock icon in the system tray. Place a check mark next to one or more options or click All ➤ Locked.

Quick Reference

Commands

CLEANSCREENON

Clears the screen of toolbars and dockable windows (excluding the command line).

CLEANSCREENOFF

Restores display of toolbars and dockable windows (excluding the command line).

DRAGMODE

Controls the way dragged objects are displayed.

HIDEPALETTES

Hides currently displayed palettes (including the command line).

LOGFILEOFF

Closes the text window log file opened by LOGFILEON.

LOGFILEON

Writes the text window contents to a file.

OPTIONS

Customizes the program settings.

SHOWPALETTES

Restores the display of hidden palettes.

VIEWRES

Sets the resolution for objects in the current viewport.

VTOPTIONS

Displays a change in view as a smooth transition.

System Variables

APERTURE

Sets the display size for the object snap target box, in pixels.

CALCINPUT

Controls whether mathematical expressions and global constants are evaluated in text and numeric entry boxes of windows and dialog boxes.

CLEANSCREENSTATE

Indicates whether the clean screen state is on or off.

CURSORSIZE

Determines the size of the crosshairs as a percentage of the screen size.

DCTCUST

Displays the path and file name of the current custom spelling dictionary.

DCTMAIN

Displays the three letter keyword for the current main spelling dictionary.

DEFPLSTYLE

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.

DRAGMODE

Controls the way dragged objects are displayed.

EXTNAMES

Sets the parameters for named object names (such as linetypes and layers) stored in definition tables.

FILLMODE

Specifies whether hatches and fills, 2D solids, and wide polylines are filled in.

GRIPBLOCK

Controls the display of grips in blocks.

GRIPCOLOR

Controls the color of unselected grips.

GRIPHOT

Controls the color of selected grips.

GRIPS

Controls the use of selection set grips for the Stretch, Move, Rotate, Scale, and Mirror Grip modes.

INSUNITS

Specifies a drawing-units value for automatic scaling of blocks, images, or xrefs when inserted or attached to a drawing.

INSUNITSDEFSOURCE

Sets source content units value when INSUNITS is set to 0.

INSUNITSDEFTARGET

Sets target drawing units value when INSUNITS is set to 0.

INTELLIGENTUPDATE

Controls the graphics refresh rate.

ISAVEBAK

Improves the speed of incremental saves, especially for large drawings.

ISAVEPERCENT

Determines the amount of wasted space tolerated in a drawing file.

LAYOUTREGENCTL

Specifies how the display list is updated in the Model tab and layout tabs.

LOCALE

Displays a code that indicates the current locale.

LOCALROOTPREFIX

Stores the full path to the root folder where local customizable files were installed.

LOCKUI

Locks the position and size of toolbars and dockable windows such as DesignCenter and the Properties palette.

LOGFILEMODE

Specifies whether the contents of the text window are written to a log file.

LOGFILENAME

Specifies the path and name of the text window log file for the current drawing.

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Specifies the path for the text window log files for all drawings in a session.

MTEXTED

Sets the application for editing multiline text objects.

OLEQUALITY

Sets the default plot quality for OLE objects.

OLESTARTUP

Controls whether the source application of an embedded OLE object loads when plotting.

OSNAPCOORD

Controls whether coordinates entered on the command line will override running object snaps.

PAPERUPDATE

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.

PALETTEOPAQUE

Controls whether palettes can be made transparent.

PICKADD

Controls whether subsequent selections replace the current selection set or add to it.

PICKAUTO

Controls automatic windowing at the Select Objects prompt.

PICKBOX

Sets the object selection target height, in pixels.

PICKDRAG

Controls the method of drawing a selection window.

PICKFIRST

Controls whether you select objects before (noun-verb selection) or after you issue a command.

PICKSTYLE

Controls the use of group selection and associative hatch selection.

PSTYLEPOLICY

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.

QTEXTMODE

Controls how text is displayed.

RASTERPREVIEW

Controls whether BMP preview images are saved with the drawing.

ROLLOVERTIPS

Controls the display of rollover tooltips in the application.

SAVEFILEPATH

Stores the current automatic save file name.

SAVEFILEPATH

Specifies the path to the directory for all automatic save files for the current session.

SAVETIME

Sets the automatic save interval, in minutes.

SPLINESEGS

Sets the number of line segments to be generated for each spline-fit polyline generated by the Spline option of the PEDIT command.

TDUSRTIMER

Stores the user-elapsed timer.

TOOLTIPMERGE

Combines drafting tooltips into a single tooltip.

TOOLTIPS

Controls the display of tooltips on the ribbon, toolbars, and other user interface elements.

VISRETAIN

Controls the properties of xref-dependent layers.

VTDURATION

Sets the duration of a smooth view transition, in milliseconds.

VTENABLE

Controls when smooth view transitions are used.

VTFPS

Sets the minimum speed of a smooth view transition, in frames per second.

XLOADCTL

Turns xref demand-loading on and off, and controls whether it opens the referenced drawing or a copy.

Utilities

No entries

Command Modifiers

No entries

Create Task-Based Workspaces

Workspaces are sets of menus, toolbars, palettes, and ribbon control panels that are grouped and organized so that you can work in a custom, task-oriented drawing environment.

When you use a workspace, only the menus, toolbars, and palettes that are relevant to a task are displayed. In addition, a workspace may automatically display the ribbon, a special palette with task-specific control panels.

For more information about using the ribbon, see The Ribbon on page 22.

You can easily switch between workspaces. The following task-based workspaces are already defined in the product:

- 2D Drafting & Annotation

- 3D Modeling
- AutoCAD Classic

For example, when you create 3D models, you can use the 3D Modeling workspace that contains only 3D-related toolbars, menus, and palettes. Interface items that you do not need for 3D modeling are hidden, maximizing the screen area available for your work.

When you make changes to your drawing display (such as moving, hiding, or displaying a toolbar or a tool palette group) and you want to preserve the display settings for future use, you can save the current settings to a workspace.

Switch Workspaces

You can switch to another workspace whenever you need to work on a different task from the Workspace icon on the status bar.

Create or Change a Workspace

You can create your own workspaces and modify the default workspaces. To create or change a workspace, use either of the following methods:

- Display, hide, and rearrange your toolbars and windows, modify your ribbon settings, and then save the current workspace from the Workspaces icon in the status bar, Workspaces toolbar or the Window menu, or use the WORKSPACE command.
- For more extensive changes, open the Customize User Interface dialog box to set up the workspace environment.

You can control the display order of your saved workspaces and other options in the Workspace Settings dialog box.

NOTE For more information about creating or modifying workspaces, and how toolbars and menus interact with workspaces, see Customize Workspaces in the *Customization Guide*.

Select a Sample Workspace

You can experiment with the sample workspace included with the product. This predefined workspace demonstrates how you might use a workspace to streamline your work tasks.

You can find the sample workspace in the following location:

<drive>:\Documents and Settings\<user name>\Application Data\Autodesk\AutoCAD 2010\<release>\<product language>\Support\acadSampleWorkspaces.cuix

To use the sample workspace, you must first transfer it to your main customization (CUIx) file. To learn more about transferring a workspace, see Transfer and Migrate Customization in the *Customization Guide*.

Workspaces and Profiles

Workspaces work with and complement the control over your drawing environment that profiles provide.

Workspaces control the display of menus, toolbars, and palettes in the drawing area. When you use or switch a workspace, you change the display of your drawing area. You manage your workspaces from the Customize User Interface dialog box.

Profiles save environment settings including many of your user options, drafting settings, paths, and other values. Profiles are updated each time you make a change to an option, setting, or other value. You can manage your profiles from the Options dialog box.

NOTE When you make changes to the drawing display, the changes are stored in your profile and are displayed the next time you launch the program, regardless of your workspace settings. The profile changes are not automatically saved to a workspace unless you select the Automatically Save Workspace Changes option in the Workspace Settings dialog box. To preserve profile settings in a workspace, click Save Current As from the shortcut menu of the workspace icon on the status bar.

For more information about profiles, see Save and Restore Interface Settings (Profiles) on page 155.

To switch workspaces

1 On the status bar, click Workspace Switching.

2 From the list of workspaces, select the workspace you want to switch to.

The workspace with a check mark is your current workspace.

Command entry: WORKSPACE

Menu: Tools ➤ Workspaces

To change workspace settings

1 Click Tools menu ➤ Workspaces ➤ Workspace Settings.

2 In the Workspace Settings dialog box, change workspace settings as needed.

3 Click OK.

Command entry: WORKSPACE, WSSETTINGS

To save a workspace

1 Tools ➤ Workspaces ➤ Save Current As.

2 In the Save Workspace dialog box, enter a name for the new workspace or select a name from the drop-down list.

3 Click Save.

Command entry: WORKSPACE, WSSAVE

Quick Reference

Commands

WORKSPACE

Creates, modifies, and saves workspaces and makes a workspace current.

WSSAVE

Saves a workspace.

WSSETTINGS

Sets options for workspaces.

System Variables

WSCURRENT

Returns the current workspace name at the Command prompt and sets a workspace to current.

Utilities

No entries

Command Modifiers

No entries

Save and Restore Interface Settings (Profiles)

Profiles store drawing environment settings. You can create profiles for different users or projects, and you can share profiles by importing and exporting them as files.

Profiles store settings such as the following:

- Default search and project file paths
- Template file locations
- Initial folder specified in file navigation dialog boxes
- Default linetype and hatch pattern files
- Printer defaults

Profile information is typically set on the Files tab of the Options dialog box, stored in the system registry, and can be exported to a text file (an ARG file).

Use Profiles for Shared Computers or Different Projects

Use the Profiles tab in the Options dialog box to create and save your drawing environment settings as a profile. If you share your workstation with other users who use the same login name, you can restore your options by making the profile current. You can also create and save profiles to use with different projects. By default, your current options are stored in a profile named <<Unnamed Profile>>. The current profile name, as well as the current drawing name, are displayed in the Options dialog box.

Transfer Profiles between Computers

Once you save a profile, you can export or import the ARG file to and from different computers. This is useful for copying, restoring, or standardizing project-oriented work environments over a network.

If you make changes to your current profile during a work session and you want to save those changes in an ARG file, you must re-export the profile.

When you export the profile with the current profile name, the ARG file is updated with the new settings.

For more information about profiles, see OPTIONS in the *Command Reference*.

Customize the Default Profile with Initial Setup

When using Initial Setup, a new profile is created based on the profile named <<Unnamed profile>>, or the current profile if <<Unnamed profile>> does not exist. The new profile created is named Initial Setup Profile, and is assigned the industry and default drawing template selected from Initial Setup. For more information on Initial Setup, see Customize AutoCAD with Initial Setup in the *Stand-Alone Installation Guide*.

To make a profile current

1. Click the Application button. At the bottom of the Application menu, click Options.
2. In the Options dialog box, Profiles tab, select the profile you want to make current.
3. Click Set Current.
4. Click OK.

Command entry: OPTIONS

To save a profile

1. Click the Application button. At the bottom of the Application menu, click Options.
2. In the Options dialog box, Profiles tab, click Add to List.
3. In the Add Profiles dialog box, enter a Profile name and Description.
4. Click Apply & Close to record the current option settings in the system registry and close the dialog box.
5. Click OK.

Command entry: OPTIONS

To make a profile current before starting the program

1. On the Windows desktop, right-click the program icon. Click Properties.

2 In the AutoCAD Properties dialog box, Shortcut tab, under Target, enter **/p currentprofile** after the current target directory. For example, to make the profile User12 current, enter the following in Target:

"c:\Program Files\<current release name>\acad.exe"/p user12

3 Click OK.

The profile name you enter is the current profile each time you start the program.

Quick Reference

Commands

OPTIONS

Customizes the program settings.

System Variables

CPROFILE

Displays the name of the current profile.

Utilities

No entries

Command Modifiers

No entries

Customize Startup

Command line switches can specify a separate startup routine for each project.

You can use command line switches to specify several options when you start the program. For example, you can run a script, start with a specified drawing template, and display a specified view when a drawing is opened. With command line switches, you can also set up several program icons, each with different start-up options.

Command line switches are parameters you can add to the *acad.exe* command line associated with a Microsoft® Windows® shortcut icon or the Windows

Run dialog box. You can include several switches within a single command line. Valid switches are listed in the following table.

/b	Script name	Designates a script to run after you start the program (b stands for batch process). Scripts can be used to set up drawing parameters in a new drawing file. An SCR file type is assumed.
/t	Template file name	Creates a new drawing based on a template or prototype drawing. A DWT file type is assumed.
/c	Configuration folder	Specifies the path for the hardware configuration file that you want to use. You can specify a directory or a particular file. A CFG file type is assumed. If you don't set the /c switch, the executable directory is searched and the ACADCFGW or ACADCFG environment variable is used as a way to define the configuration file and directory location.
/v	View name	Designates a particular view of the drawing for display at startup.
/ld	ARX or DBX application	Loads a specified ARX or DBX application. Use the following format: <path>\<filename>.ARX If the path or file name contains spaces, then the path or file name should be wrapped in double quotes. If no path information is included, the program search path is used.
/s	Support folders	Designates support folders other than the current folder. Drawing support files include text fonts, menus, AutoLISP files, linetypes, and hatch patterns. The maximum number of folders you can specify in the path is 15. Each folder name is delimited by semicolons.
/r	Default system pointing device	Restores the default system pointing device. It creates a new configuration file

		(*acad2010.cfg*) and renames the previous configuration file to *acad.bak*.
/nologo	No AutoCAD logo screen	Starts the program without first displaying the logo screen.
/p	User-defined registry profile for starting the program	Specifies a user-defined registry profile for starting the program. The selected profile is in effect only for the current session of the program, unless you make another profile current in the Options dialog box during that session. You create or import profiles on the Profiles tab in the Options dialog box. With the /p switch, you can specify only those profiles that are listed in the Options dialog box. If the profile does not exist, the current profile is used.
/nohard-ware	Disables hardware acceleration	Disables hardware acceleration on startup.
/nossm	No Sheet Set Manager window	Suppresses the display of the Sheet Set Manager window on startup.
/set	Sheet set	Loads the named sheet set on startup. Use the following format: <path>\<sheet set data file>.DST
/w	Default workspace	Designates which workspace in the loaded CUIx files should be restored on startup.
/pl	Background plotting/publish-ing	Publishes a drawing set descriptions (DSD) file in the background. Use the following format: <path>\<drawing set descriptions file>.DSD

The syntax for using command line switches is

"*drive*:*pathname**acad.exe*" ["*drawingname*"] [/*switch* "*name*"]

When using a switch option, you must follow the switch with a space and then the name of a file, path, or view within quotation marks. For example, the following entry starts the program from a folder named *AutoCAD 2010* with the drawing template *arch1.dwt*, restores a named view PLAN1, and executes a script file *startup.scr*.

"d:\AutoCAD 2010*acad.exe*"/t "d:\AutoCAD 2010\template\arch1" /v "plan1" /b "startup"

The environment settings are resolved in the following way:

- If you use a command line switch to specify an environment setting, the command line switch overrides the settings specified in either the Options dialog box or the environment variable.
- If a command line switch is not set, the corresponding value set in the Options dialog box is used.
- If neither a command line switch nor an Options value is set, the environment variable value is used.

NOTE Command line switches and environment variables override values set in the Options dialog box for the current session only. They do not alter the system registry.

To start the program with a command line switch

1 Right-click the program icon on the Windows desktop. Click Properties.

2 In the AutoCAD Properties dialog box, Shortcut tab, in the Target box, edit the parameters for the switch using the following syntax:

"*drive*:*pathname**acad.exe*" ["*drawing name*"] [/*switch* "*name*"]

Valid switches are as follows:

/b	Script name (*b* stands for batch process)
/t	Template file name
/c	Configuration folder
/v	View name
/s	Support folders
/r	Default system pointing device
/nologo	No AutoCAD logo screen

/nohardware	Disables hardware acceleration
/p	User-defined registry profile
/nossm	No Sheet Set Manager window
/set	Sheet Set name
/w	Default workspace
/pl	Background plotting/publishing for a Drawing Set Descriptions (DSD) file

For example, enter "**d:**\AutoCAD 2010*acad.exe*" **/t** "**d:**\AutoCAD 2010**template****arch1**" **/v** "**plan1**" **/b** "**startup**"

3 Click OK.

Quick Reference

Commands

No entries

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Play Back an Action Macro

After an action macro is recorded with the Action Recorder, you can play back the series of recorded commands and input values.

You play back an action macro from the Action Recorder panel. You can also enter the name of the action macro at the Command prompt. As an action macro is played back, you might be paused for input or requested to respond to a message or requested to insert a base point.

Based on the current action or request for user interaction in the action macro, an icon near the cursor is displayed to indicate when the action macro needs input in order to continue. A dialog box may be displayed where you enter a value or use the recorded value.

As an action macro is played back, the series of commands are performed one after the other until playback is complete or an error is encountered. Some of the reasons for the playback to stop or fail are as follows:

- **Invalid command.** The command that is defined in the action macro is unknown to AutoCAD. The action macro might have been recorded in a different product or contains custom commands or macros that are not loaded.
- **Empty selection.** The current action expected a selection set of objects, but no objects were selected.
- **Macro cancelled.** The Cancel button in one of the dialogs boxes was clicked.

Location of Action Macros

When you record an action macro, it is saved to the path defined by the system variable ACTRECPATH. For playback, paths are defined by the system variable ACTPATH.

Both sets of paths are used when loading and playing back an action macro. In the Action Recorder Settings node on the Files tab of the Options dialog box, you can to set the paths used for both recording and reading action macro files.

Record and Modify Action Macros

Recording and modifying action macros is done with the Action Recorder panel on the ribbon. To learn how to record and modify action macros, see Record and Modify Action Macros in the Customization Guide.

See also:

- Record and Modify Action Macros

To play back an action macro from the Action Recorder panel

1 On the ribbon, click Manage tab ➤ Action Recorder panel. Click the down arrow next to the Action Macro list and select the action macro to play back.

2 Click Play.

3 Follow any Command prompts and dialog boxes displayed to complete the action macro.

To play back an action macro from the Command prompt

- At the Command prompt, enter the name of the action macro that you want to play back and press Enter.
  For example, if the action macro command name is DRAWLINE, enter `drawline` at the Command prompt and press Enter.

To provide user input during playback

1 Start the playback on an action macro.

2 Follow the prompts displayed at the Command prompt.

To change the location used to search for recorded action macros

1 Click the Application button. At the bottom of the Application menu, click Options.

2 In the Options dialog box, Files tab, in the list of nodes, click the plus sign (+) next to Action Recorder Settings.

3 Click the plus sign (+) next to Additional Actions Reading File Locations, and specify a folder path.

4 Click OK

Command entry: OPTIONS
Shortcut menu: Right-click in the drawing window. Click Options.

Quick Reference

Commands

OPTIONS

Customizes the program settings.

RIBBON

Opens the ribbon window.

System Variables

ACTPATH

Specifies the additional paths to use when locating available action macros for playback.

ACTRECPATH

Specifies the path used to store new action macros.

Utilities

No entries

Command Modifiers

No entries

Migrate and Specify Initial Setup

Migrating from an older release or getting started with AutoCAD for the first time present different challenges.

Migrate Custom Settings allows you to copy your custom settings and files from a previous release of AutoCAD to the latest release, making it easier to start using the new release. Initial Setup helps you to perform some basic customization of AutoCAD the first time you start it.

Migrate Custom Settings and Files

One of the challenges when moving from a previous release of AutoCAD to the latest release is getting the new release to look and behave like the previous release. You can use Migrate Custom Settings and the Customize User Interface (CUI) Editor to transfer many of your custom settings and files to the latest release.

Initial Setup

You can perform some basic customization of AutoCAD with Initial Setup which makes it easier to access specific tools and start a new drawing. Initial Setup is displayed the first time you start AutoCAD and can also be accessed from the Options dialog box. The following can be done with Initial Setup:

- Specify an industry that best describes your work to help search for related content to use in drawings and companion products developed by partners.
- Add task-based tools to your default workspace.
- Specify a drawing template to use when creating new drawings.

For more information on Initial Setup, see Customize AutoCAD with Initial Setup in the *Stand-Alone Installation Guide*.

See also:

- Migrate Custom Settings and Files from Previous Releases in the *Stand-Alone Installation Guide*
- Basic Customization in the *Customization Guide*
- User Interface Customization in the *Customization Guide*

Start, Organize, and Save a Drawing

Start a Drawing

5

You can start a new drawing several ways, including starting from scratch or using a template file. In each case, you can choose the units of measurement and other unit format conventions.

Start a Drawing from Scratch

You can start from scratch from the Create New Drawing dialog box, from the Select Template dialog box, or with a default drawing template file that uses no dialog box at all.

Use a Drawing Template File

You can start a drawing that uses predefined settings from a default drawing template file.

When you use a default drawing template file, new drawings automatically use the settings defined in the specified file. That template file displays each time you start a new drawing using the QNEW command.

You can specify the location of the default drawing template file using the Options dialog box.

See also:

- Use a Template File to Start a Drawing on page 175

To specify a path for the default drawing template file

1. Click the Application button. At the bottom of the Application menu, click Options.
2. In the Options dialog box, click the Files tab.

3 On the Files tab, expand the Template Settings node.

4 In the Template Settings node, expand the Drawing Template File Location node.

5 Specify the path where your desired default drawing template file is located on your local drive.

To start a drawing from scratch from a default drawing template file

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NOTE If necessary, at the Command prompt, enter **startup** and **0**; then enter **filedia** and **1**.

---

1 Click the Application button. At the bottom of the Application menu, click Options.

2 In the Options dialog box, click the Files tab.

3 On the Files tab, expand the Template Settings node.

4 In the Template Settings node, expand the Default Template File Name for QNEW node.

5 Specify a drawing template path and file name.

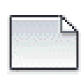
   NOTE The drawing template path and file name must end with *.dwt*.

6 Click OK.

7 At the Command prompt, enter **qnew**.

   The new drawing starts as *drawing1.dwg*. The default drawing name changes to reflect the number of new drawings that have been started. For example, if you start another drawing, the default drawing name is *drawing2.dwg*.

Command entry: QNEW

Toolbar: Standard 

Quick Reference

Commands

NEW

Creates a new drawing.

OPTIONS

Customizes the program settings.

QNEW

Starts a new drawing with a selected drawing template file.

System Variables

FILEDIA

Suppresses display of file navigation dialog boxes.

MEASUREINIT

Controls whether a drawing you start from scratch uses imperial or metric default settings.

MEASUREMENT

Controls whether the current drawing uses imperial or metric hatch pattern and linetype files.

STARTUP

Controls whether the Create New Drawing dialog box is displayed when a new drawing is started with NEW or QNEW.

Utilities

No entries

Command Modifiers

No entries

Define Settings When You Create A New Drawing

You can define drawing settings each time you create a new drawing with the Create New Drawing dialog box.

You can use the Create New Drawing dialog box to choose either imperial or metric units for the new drawing. The setting you select determines default values used for many system variables controlling text, dimensions, grid, snap, and the default linetype and hatch pattern file.

- **Imperial.** Creates a new drawing based on the imperial measurement system. The drawing uses internal default values, and the default grid display boundary, called the *gridlimits*, is 12 x 9 inches.
- **Metric.** Creates a new drawing based on the metric measurement system. The drawing uses internal default values, and the default grid display boundary is 420 x 290 millimeters.

To start a drawing from scratch with the Create Drawing dialog box

---

NOTE If necessary, at the Command prompt, enter **startup** and **1**; then enter **filedia** and **1**.

---

1 Click File menu ➤ New..

2 In the Create New Drawing dialog box, click Start from Scratch.

3 Under Default Settings, click either Imperial or Metric.

   The new drawing starts as *drawing1.dwg*. The default drawing name changes to reflect the number of new drawings that have been started. For example, if you start another drawing, the default drawing name is *drawing2.dwg*.

Command entry: NEW

Quick Reference

Commands

NEW

Creates a new drawing.

OPTIONS

Customizes the program settings.

QNEW

Starts a new drawing with a selected drawing template file.

System Variables

FILEDIA

Suppresses display of file navigation dialog boxes.

MEASUREINIT

Controls whether a drawing you start from scratch uses imperial or metric default settings.

MEASUREMENT

Controls whether the current drawing uses imperial or metric hatch pattern and linetype files.

STARTUP

Controls whether the Create New Drawing dialog box is displayed when a new drawing is started with NEW or QNEW.

Utilities

No entries

Command Modifiers

No entries

Select a Template When You Create a New Drawing

You can choose a different drawing template file each time you create a new drawing with the Select Template dialog box.

If you require different settings each time you create a new drawing, such as an imperial template file and a metric template file, you can use the Select Template dialog box to create a new drawing.

NOTE At the bottom-right corner of the Select Template dialog box, there is an Open button with an arrow button next to it. If you click the arrow button, you can choose between two internal default drawing templates, metric or imperial.

See also:

- Use a Template File to Start a Drawing on page 175

To choose a drawing template file each time you create a new drawing

---

NOTE If necessary, at the Command prompt, enter **startup** and **0**; then enter **filedia** and **1**.

---

1. Click the Application button. At the bottom of the Application menu, click Options.
2. In the Options dialog box, click the Files tab.
3. On the Files tab, expand the Template Settings node.
4. In the Template Settings node, expand the Default Template File Name for QNEW node and double-click the contents.
5. Specify a drawing template folder path.
6. Click OK.
7. Click the Application button, and click New menu ➤ Drawing.

   The new drawing starts as *drawing1.dwg*. The default drawing name changes to reflect the number of new drawings that have been started. For example, if you start another drawing, the default drawing name is *drawing2.dwg*.

   QNEW

Quick Reference

Commands

NEW

Creates a new drawing.

OPTIONS

Customizes the program settings.

QNEW

Starts a new drawing with a selected drawing template file.

System Variables

FILEDIA

Suppresses display of file navigation dialog boxes.

MEASUREINIT

Controls whether a drawing you start from scratch uses imperial or metric default settings.

MEASUREMENT

Controls whether the current drawing uses imperial or metric hatch pattern and linetype files.

STARTUP

Controls whether the Create New Drawing dialog box is displayed when a new drawing is started with NEW or QNEW.

Utilities

No entries

Command Modifiers

No entries

Use a Template File to Start a Drawing

A drawing template file is used to provide consistency in the drawings that you create by providing standard styles and settings.

Drawing template files have a *.dwt* file extension.

When you create a new drawing based on an existing drawing template file and make changes, the changes in the new drawing do not affect the drawing template file.

Select a Drawing Template File

A default set of drawing template (DWT) files is installed with AutoCAD which are used to create 2D drawings and 3D models. Many of the default drawing templates are offered in two measurement types: imperial and metric. When using one of the default drawing templates you need determine if you will be working in 2D or 3D, and which measurement type best describes your work.

While the default templates provide for a quick way to start creating a new drawing, it is best to create drawing templates specific to your company and the drawings you create. You can use Initial Setup to select a drawing template that is close to your industry, but you will still need to do some work to make the drawing template file work best for you. For more information on Initial

Setup, see Customize AutoCAD with Initial Setup in the *Stand-Alone Installation Guide*.

Create a Drawing Template File

When you need to create several drawings that use the same conventions and default settings, you can save time by creating or customizing a drawing template file instead of specifying the conventions and default settings each time you start. Conventions and settings commonly stored in template files include

- Unit type and precision
- Title blocks, borders, and logos
- Layer names
- Snap, Grid, and Ortho settings
- Grid limits
- Annotation styles (dimension, text, table and multileader)
- Linetypes

By default, drawing template files are stored in the *template* folder, where they are easily accessible.

Recover the Default Drawing Template File

If the settings in the drawing template file *acad.dwt* or *acadiso.dwt* have been changed from the original defaults, you can reset them by starting a new drawing with no template and then saving the drawing as a drawing template file, replacing *acad.dwt* or *acadiso.dwt*.

If you specify the 3D Modeling workspace, the default drawing template files are *acad3d.dwt* and *acadiso3d.dwt*.

You can start a new drawing with the original defaults by using NEW to display the Select Template dialog box. To do this, click the arrow next to the Open button and then click one of the "no template" options from the list.

See also:

- Use a Hyperlink to Start a New Drawing

To start a drawing by selecting a template file

1 Click the Application menu, and click New menu ➤ Drawing.

2 In the Select Template dialog box, select a template from the list.

3 Click Open.

The drawing is opened as *drawing1.dwg*. The default drawing name changes to reflect the number of new drawings that have been opened. For example, if you open another drawing from a template, the default drawing name is *drawing2.dwg*.

To start a new drawing with no template file, click the arrow next to the Open button. Select one of the "no template" options from the list.

Menu: File ➤ New
Command entry: NEW

To create a drawing template file from an existing drawing

1 Click the Application button, and click Open ➤ Drawing.

2 In the Select File dialog box, select the file you want to use as a template. Click OK.

3 To delete the existing file contents, click Home tab ➤ Modify panel ➤ Erase.

4 At the Select Objects prompt, enter **all**, and then select the border and title block (if you want to remove them) and enter **r** (Remove).

5 Click the Application button, and click Save As ➤ AutoCAD Drawing Template.

DWT files must be saved in the current drawing file format. To create a DWT file in a previous format, save the file in the desired DWG format, and then rename the DWG file using a DWT extension.

6 In the Save Drawing As dialog box, File Name text box, enter a name for the template.

7 Click Save.

8 Enter a description of the template.

9 Click OK.

The new template is saved in the *template* folder.

Command entry: OPEN, SAVEAS

Toolbar: Standard 

To recover the default drawing template files

1 Click the Application button, and click New menu ➤ Drawing.

2 In the Select Template dialog box, click the arrow next to the Open button. Select one of the options from the list:

- Open with No Template - Imperial (to restore *acad.dwt)*
- Open with No Template - Metric (to restore *acadiso.dwt)*

The drawing opens with the default settings.

3 In the Save Drawing As dialog box, save the drawing with its original name, either *acad.dwt* for imperial or *acadiso.dwt* for metric.

4 Click Save.

NOTE For the 3D Modeling workspace, the default drawing template files are *acad3d.dwt* and *acadiso3d.dwt*

Quick Reference

Commands

NEW

Creates a new drawing.

OPEN

Opens an existing drawing file.

OPTIONS

Customizes the program settings.

SAVEAS

Saves a copy of the current drawing under a new file name.

System Variables

MEASUREMENT

Controls whether the current drawing uses imperial or metric hatch pattern and linetype files.

Utilities

No entries

Command Modifiers

No entries

Use a Wizard to Start a Drawing

A setup wizard establishes basic drawing settings step by step.

You have two wizard options to help you set up a drawing:

- **Quick Setup Wizard.** Sets units of measurement, precision of displayed units, and grid limits.
- **Advanced Setup Wizard.** Sets units of measurement, precision of displayed units, and grid limits. Also establishes angle settings such as units of measurement style, precision, direction, and orientation.

These wizards are available in the Create New Drawing dialog box.

To begin a new drawing using a wizard

1. If necessary, set the STARTUP system variable to 1 and the FILEDIA system variable to 1.
2. Click File menu ➤ New.
3. In the Create New Drawing dialog box, click Use a Wizard.
4. Click Quick Setup or Advanced Setup.
5. Complete the wizard pages using the Next and Back buttons to move forward and backward.
6. On the last page, click Finish.

Command entry: NEW

Quick Reference

Commands

NEW

Creates a new drawing.

System Variables

MEASUREINIT

Controls whether a drawing you start from scratch uses imperial or metric default settings.

MEASUREMENT

Controls whether the current drawing uses imperial or metric hatch pattern and linetype files.

Utilities

No entries

Command Modifiers

No entries

Specify Units and Unit Formats

Before you start to draw, you decide on the units of measurement to be used in the drawing, and set the format, precision, and other conventions to be used in coordinates and distances.

Determine the Units of Measurement

Before you start to draw, you must decide what one drawing unit represents based on what you plan to draw. You can convert a drawing between systems of measurement by scaling it.

Every object you create is measured in drawing units. Before you start to draw, you must decide what one drawing unit will represent based on what you plan to draw. Then you create your drawing at actual size with that convention. For example, a distance of one drawing unit typically represents one millimeter, one centimeter, one inch, or one foot in real-world units.

Convert Drawing Units

If you start a drawing in one system of measurement (imperial or metric) and then want to switch to the other system, use SCALE to scale the model geometry by the appropriate conversion factor to obtain correct distances and dimensions.

For example, to convert a drawing created in inches to centimeters, you scale the model geometry by a factor of 2.54. To convert from centimeters to inches, the scale factor is 1/2.54 or about 0.3937.

See also:

- Set the Scale for Dimensions on page 1673

To set the units format and precision

1. Click the Application button, and click Drawing ➤ Units.
2. In the Drawing Units dialog box, under Length, select a unit format and precision. Click OK.

   The Sample Output area shows an example of the unit format at the current precision.

Command entry: UNITS

To convert a drawing from inches to centimeters

1. Click Home tab ➤ Modify panel ➤ Scale. 
2. At the Select Objects prompt, enter **all**.

   All objects in the drawing are selected for scaling.
3. Enter a base point of ***0,0**.

   Scaling will be relative to the world coordinate system origin and the location of the drawing origin will remain at the WCS origin.
4. Enter a scale factor of **2.54** (there are 2.54 centimeters per inch).

   All objects in the drawing are now 2.54 times larger, corresponding to the equivalent distance in centimeters.

Command entry: SCALE

Toolbar: Modify

To convert a drawing from centimeters to inches

1 Click Home tab ➤ Modify panel ➤ Scale.

2 At the Select Objects prompt, enter **all**.

All objects in the drawing are selected for scaling.

3 Enter a base point of ***0,0**.

Scaling is performed relative to the world coordinate system origin, and the location of the drawing origin will remain at the WCS origin.

4 Enter a scale factor of 0.3937 (the inverse of 2.54 centimeters per inch).

All objects in the drawing are now smaller, corresponding to the equivalent distance in inches.

Toolbar: Modify
Command entry: SCALE

Quick Reference

Commands

UNITS

Controls coordinate and angle display formats and precision.

SCALE

Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling.

System Variables

LUNITS

Sets linear units.

LUPREC

Sets the display precision for linear units and coordinates.

MEASUREINIT

Controls whether a drawing you start from scratch uses imperial or metric default settings.

MEASUREMENT

Controls whether the current drawing uses imperial or metric hatch pattern and linetype files.

UNITMODE

Controls the display format for units.

Utilities

No entries

Command Modifiers

No entries

Set Linear Unit Conventions

Before you start to draw, you set the format and the number of decimal places to be used when you enter and display linear units.

You can choose from several common conventions to represent the display style and the precision of distances and coordinates. Depending on what you specify, you can enter and display in decimal format, fractional format, or other notation. These settings affect

- The Properties palette
- Dynamic input
- The LIST command
- The ID command
- The coordinate display on the status bar
- Several dialog boxes that display coordinates

You can set the unit type and precision in the Drawing Units dialog box, the Quick Setup wizard, or the Advanced Setup wizard.

Understand Rounding and Precision

When you specify the display precision of units, the values for coordinates and distances are rounded off. However, the internal precision of coordinates and distances is always maintained regardless of the display precision.

For example, if you set the display precision of decimal-format units to 1 (or 0.0), the display of coordinates is rounded to one place after the decimal point. Thus, the coordinates 0.000,1.375 are displayed as 0.0,1.4, but the internal precision is still maintained.

Enter Distances in Imperial Architectural Format

In architectural format, to enter feet and inches, indicate feet using the prime symbol ('), for example, 72'3. You don't need to enter quotation marks (“) to specify inches.

NOTE The units format for creating and listing objects, measuring distances, and displaying coordinate locations is separate from the dimension units setting used in creating dimension values.

See also:

- Set the Scale for Dimensions on page 1673

To set the units format and precision

1. Click the Application button, and click Drawing ➤ Units.
2. In the Drawing Units dialog box, under Length, select a unit format and precision. Click OK.

   The Sample Output area shows an example of the unit format at the current precision.

Command entry: UNITS

To convert a drawing from inches to centimeters

1. Click Home tab ➤ Modify panel ➤ Scale. 

2. At the Select Objects prompt, enter **all**.

   All objects in the drawing are selected for scaling.

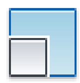
3. Enter a base point of ***0,0**.

   Scaling will be relative to the world coordinate system origin and the location of the drawing origin will remain at the WCS origin.

4. Enter a scale factor of **2.54** (there are 2.54 centimeters per inch).

   All objects in the drawing are now 2.54 times larger, corresponding to the equivalent distance in centimeters.

Command entry: SCALE

Toolbar: Modify 

To convert a drawing from centimeters to inches

1. Click Home tab ➤ Modify panel ➤ Scale. 
2. At the Select Objects prompt, enter **all**.

   All objects in the drawing are selected for scaling.

3. Enter a base point of ***0,0**.

   Scaling is performed relative to the world coordinate system origin, and the location of the drawing origin will remain at the WCS origin.

4. Enter a scale factor of 0.3937 (the inverse of 2.54 centimeters per inch).

   All objects in the drawing are now smaller, corresponding to the equivalent distance in inches.

Toolbar: Modify 

Command entry: SCALE

Quick Reference

Commands

UNITS

Controls coordinate and angle display formats and precision.

System Variables

LUNITS

Sets linear units.

LUPREC

Sets the display precision for linear units and coordinates.

MEASUREINIT

Controls whether a drawing you start from scratch uses imperial or metric default settings.

MEASUREMENT

Controls whether the current drawing uses imperial or metric hatch pattern and linetype files.

UNITMODE

Controls the display format for units.

Utilities

No entries

Command Modifiers

No entries

Set Angular Unit Conventions

Angle conventions include the position of angle 0 and the direction of measurement: clockwise or counterclockwise. You also set the format and the number of decimal places.

You can choose from several common conventions to represent angles in a drawing. You can specify that positive values of angles are measured either clockwise or counterclockwise, and angle 0 can be set to any direction (usually

East or North). You can enter angles using grads, radians, or surveyor's units or using degrees, minutes, and seconds.

Work with Surveyor's Angles

If you use surveyor's angles when specifying polar coordinates, indicate whether the surveyor's angles are in the north, south, east, or west direction. For example, to enter a coordinate relative to the current coordinate for a property line that is 72 feet, 8 inches long with a bearing of 45 degrees north, 20 minutes, 6 seconds east, enter

@72'8"<n45d20'6"e

To set the angle format and precision

1. Click the Application button, and click Drawing Utilities ➤ Units.
2. In the Drawing Units dialog box, under Angle, select an angle type and precision.

   The Sample Output area shows an example of the angle type at the current precision.
3. Select Direction to specify an angle direction.

   The angle direction controls the point from which angles are measured and the direction in which they're measured. The default starting angle, 0 degrees, is toward 3 o'clock (or east), and positive angle measurement is counterclockwise.
4. Select the options you want to use.
5. Click OK to close each dialog box.

Command entry: UNITS

Quick Reference

Commands

UNITS

Controls coordinate and angle display formats and precision.

System Variables

ANGBASE

Sets the base angle to 0 with respect to the current UCS.

ANGDIR

Sets the direction of positive angles.

AUNITS

Sets units for angles.

AUPREC

Sets the display precision for angular units and coordinates.

UNITMODE

Controls the display format for units.

Utilities

No entries

Command Modifiers

No entries

Add Identifying Information to Drawings

You can keep track of your drawings more easily if you add keywords or other information to them.

Use Windows Explorer

Drawing properties can help you identify a drawing. Some drawing properties are stored by the operating system; for example, drawing type, location, and size. These values are read-only in the drawing file and can only be changed through Windows Explorer.

Use the Find Tool

The Find tool in a standard file selection dialog box can use this information. For example, you can search for all files created on a certain date, or for files you modified yesterday.

Use DesignCenter

You can create additional properties in drawing files. You can store author, title, and subject, and you can assign keywords, hyperlink addresses or directory paths, and custom properties to your drawings. The Advanced tab in the Search tool in DesignCenter can use these properties to locate drawing files. For more information about DesignCenter, see ADCENTER and Access Content with DesignCenter on page 92.

Use Sheet Set Manager Properties

With the Sheet Set Manager, you can assign a sheet title, number, and a description to every sheet in a sheet set. For more information about sheet sets, see Include Information with Sheets and Sheet Sets on page 506.

Display Properties in Fields

You can assign any of the drawing properties to a field in a text object. For more information about fields, see Use Fields in Text on page 1532.

To display drawing properties for the active drawing

1 Click the Application button, and click Drawing ➤ Drawing Properties.

2 In the Drawing Properties dialog box, click tabs to view the different types of information.

Command entry: DWGPROPS

To define drawing properties

1 Click the Application button, and click Drawing ➤ Drawing Properties.

2 In the Drawing Properties dialog box, enter property information on the following tabs:

- **Summary tab**. Enter the drawing title, subject, author, keywords, comments, and a default address for hyperlinked data in your drawing. For example, you can add the keyword *Autodesk* to certain drawing files and then use DesignCenter to search for all drawing files with that keyword. For a hyperlink base, you can specify an Internet address or a path to a folder on a network drive.

- **Custom tab.** Click Add. In the Add Custom Property dialog box, enter a name and a value for the custom property. Click OK. The new custom property and its value are displayed on the Custom tab. This information can be used for advanced searches in DesignCenter.

3 Click OK.

Command entry: DWGPROPS

Quick Reference

Commands

DWGPROPS

Sets and displays the file properties of the current drawing.

System Variables

CDATE

Stores the current date and time in decimal format.

DATE

Stores the current date and time in Modified Julian Date format.

TDCREATE

Stores the local time and date the drawing was created.

TDINDWG

Stores the total editing time, which is the total elapsed time between saves of the current drawing.

TDUCREATE

Stores the universal time and date that the drawing was created.

TDUPDATE

Stores the local time and date of the last update/save.

TDUUPDATE

Stores the universal time and date of the last update or save.

Utilities

No entries

Command Modifiers

No entries

Insert Geographical Location Information in a Drawing

You can embed the geographic location of the geometry in a drawing file.

Overview of Geographic Location

Geographic location embeds location-specific references – expressed as real-world coordinates (*X*,*Y*, and *Z*) – in your drawing.

You can then send your georeferenced drawing for review.

For example, you can:

- Put the drawing on a map (using AutoCAD Map 3D)
- See your design in the landscape (using AutoCAD)

When you add a geographic location to a drawing, a geographic marker is created.

The geographic marker is a visual representation of the location information and is created at the specified point on the drawing.

Geographic location information can be included in either of the following ways:

- Import a KML or KMZ file with the appropriate location information
- Import a location from Google Earth
- Use the Geographic Location dialog box

When you insert location information, the drawing contains the following data:

- North direction — a vector that defines the direction of the North pole from the *XY* plane

- Up direction — a vector that is always constrained 90 degrees to the *XY* plane
- Geographic location data

See also:

- Define the Geographic Location for a DWG File on page 195
- View the Geographic Location Information on page 199

To import a kml or kmz file into the drawing

1 Click Render tab ➤ Sun & Location panel ➤ Set Location. 

2 Click Import a KML or KMZ File.

3 Navigate to the location of the KML or KMZ file. Click Open.

> **NOTE** If your KML or KMZ file references multiple locations, only the first location is used. In such cases, click Close when the Multiple Locations Found dialog box is displayed.

4 Click or enter a point or value for the location in the World Coordinate System (WCS) *X*, *Y*, *Z* format.

5 Click to specify the north direction.

A geographic marker (visual representation of the location information) is inserted at the specified location.

> **NOTE** The geographic marker displays differently in 2D and 3D views.

To import the current location from Google Earth

> **NOTE** Before you proceed, you must have Google Earth installed and open, with the location selected.

1 Click Render tab ➤ Sun & Location panel ➤ Set Location. 

2 Click Import the current location from Google Earth.

The Import from Google Earth task dialog box is displayed.

3 Click Continue.

4 Click or enter a point / value for the location in the World Coordinate System (WCS) X, Y, Z format.

5 Click to specify the north direction.

The geographic marker is created at the specified point on the drawing.

NOTE The geographic marker displays differently in 2D and 3D views.

To enter the location values manually

1 Click Render tab ➤ Sun & Location panel ➤ Set Location. 

2 Click Enter the Location Values.

3 (Optional) Select the latitude and longitude format.

You can enter the latitude, longitude, and time zone values manually, or click the Use Map button (steps 4 through 8) to select the values visually.

4 (Optional) Click Use Map.

5 In the Region drop-down list, select the applicable region.

6 In the Nearest City drop-down list, select the nearest city that represents the time zone. Click OK.

NOTE The Time Zone value is updated automatically based on the Nearest City selected.

---

7 Click Accept Updated Time Zone, if correct. To select a different time zone, click Return to the Previous Dialog Box.

NOTE The latitude, longitude, direction, and time zone values are automatically populated based on the values selected in the location picker dialog box.

---

8 Click Pick Point to specify the *X*, *Y*, and *Z* coordinates. Values can also be entered at the Command prompt.

9 Use the Up or Down arrows to specify the elevation.

10 Click Pick Point, or drag the compass needle icon, to specify the angle.

The north direction angle is calculated when you select a point with reference to the geographic location.

11 Specify the Up direction, if necessary.

12 Click OK.

NOTE The geographic marker displays differently in 2D and 3D views.

To view the graphic of a geographic marker

1 At the Command prompt, enter **geomarkervisibility**. Enter 1 (Geographic marker is visible).

2 Click Render tab ➤ Sun & Location panel ➤ Set Location. 

3 Insert a geographic marker using the options available.

To view the status bar coordinate display

1 Click Render tab ➤ Sun & Location panel ➤ Set Location. 

2 Insert a geographic marker using the options available.

3 Right-click the status bar display. Click Geographic.

The status bar coordinates display the latitude and longitude values of the geographic location.

Quick Reference

Command

GEOGRAPHICLOCATION

Specifies the geographic location information for a drawing file.

INSERT

Inserts a block or drawing into the current drawing.

XATTACH

Inserts a DWG file as an external reference (xref).

System Variables

GEOLATLONGFORMAT

Controls the format of the latitude or longitude values in the Geographic Location dialog box, and the coordinate status bar in Geographic mode.

GEOMARKERVISIBILITY

Controls the visibility of geographic markers.

Define the Geographic Location for a DWG File

You can insert geographic location information in the drawing by importing .kml files, or .kmz files; or location information from Google Earth; or by manually entering the location values.

Import KML or KMZ files

You can import the location information (latitude, longitude, and altitude) specified in KML or KMZ files.

Once imported, specify the location in the drawing; and the north direction, or angle.

NOTE When a KML or KMZ file references multiple locations only the first placemark found is used.

Import Location from Google Earth

You can navigate to a specific location in Google Earth and import the location information in the drawing file.

NOTE Before proceeding, you must have Google Earth installed and open with the location selected.

You can then pick a point in the drawing to define the coordinates for the location defined in Google Earth.

Use the Geographic Location Dialog Box

You can manually enter the latitude, longitude, north direction, elevation, and up-direction values.

You can also enter a city name or other known geographic entity in the Location Picker dialog box to insert the location.

Latitude and longitude is defined either as Decimal Degrees or as Degree Minutes Second.

To import a kml or kmz file into the drawing

1. Click Render tab ➤ Sun & Location panel ➤ Set Location. 
2. Click Import a KML or KMZ File.
3. Navigate to the location of the KML or KMZ file. Click Open.

   NOTE If your KML or KMZ file references multiple locations, only the first location is used. In such cases, click Close when the Multiple Locations Found dialog box is displayed.

4. Click or enter a point or value for the location in the World Coordinate System (WCS) *X*, *Y*, *Z* format.
5. Click to specify the north direction.

   A geographic marker (visual representation of the location information) is inserted at the specified location.

   NOTE The geographic marker displays differently in 2D and 3D views.

To import the current location from Google Earth

NOTE Before you proceed, you must have Google Earth installed and open, with the location selected.

1. Click Render tab ➤ Sun & Location panel ➤ Set Location. 
2. Click Import the current location from Google Earth.

   The Import from Google Earth task dialog box is displayed.

3. Click Continue.
4. Click or enter a point / value for the location in the World Coordinate System (WCS) X, Y, Z format.

5 Click to specify the north direction.

The geographic marker is created at the specified point on the drawing.

NOTE The geographic marker displays differently in 2D and 3D views.

To enter the location values manually

1 Click Render tab ➤ Sun & Location panel ➤ Set Location. 

2 Click Enter the Location Values.

3 (Optional) Select the latitude and longitude format.

You can enter the latitude, longitude, and time zone values manually, or click the Use Map button (steps 4 through 8) to select the values visually.

4 (Optional) Click Use Map.

5 In the Region drop-down list, select the applicable region.

6 In the Nearest City drop-down list, select the nearest city that represents the time zone. Click OK.

NOTE The Time Zone value is updated automatically based on the Nearest City selected.

---

7 Click Accept Updated Time Zone, if correct. To select a different time zone, click Return to the Previous Dialog Box.

NOTE The latitude, longitude, direction, and time zone values are automatically populated based on the values selected in the location picker dialog box.

---

8 Click Pick Point to specify the *X*, *Y*, and *Z* coordinates. Values can also be entered at the Command prompt.

9 Use the Up or Down arrows to specify the elevation.

10 Click Pick Point, or drag the compass needle icon, to specify the angle.

The north direction angle is calculated when you select a point with reference to the geographic location.

11 Specify the Up direction, if necessary.

12 Click OK.

NOTE The geographic marker displays differently in 2D and 3D views.

To edit a geographic marker

1 Click Render tab ➤ Sun & Location panel ➤ Set Location. 

2 Click Edit Current Geographic Location.

3 Edit the existing values you want to change. Click OK.

To remove a geographic marker

1 Click Render tab ➤ Sun & Location panel ➤ Set Location. 

2 Click Remove Geographic Location.

3 In the Confirmation dialog box, click Yes.

To import a drawing with an external coordinate system defined

1 Click Render tab ➤ Sun & Location panel ➤ Set Location. 

2 Click one of the following options:

- Leave Existing Coordinate System —AutoCAD does not convert the external coordinate system to the LL84 coordinate system.

- Convert to AutoCAD and then edit — Converts the coordinate system to theLL84 system and then displays the Location Already Exists task dialog box.

3 Edit the geographic location using the options presented.

Quick Reference

Command

GEOGRAPHICLOCATION

Specifies the geographic location information for a drawing file.

INSERT

Inserts a block or drawing into the current drawing.

XATTACH

Inserts a DWG file as an external reference (xref).

System Variables

GEOLATLONGFORMAT

Controls the format of the latitude or longitude values in the Geographic Location dialog box, and the coordinate status bar in Geographic mode.

GEOMARKERVISIBILITY

Controls the visibility of geographic markers.

View the Geographic Location Information

You can view a marker that defines the geographic location as a graphic (in model space view) or as a status bar coordinate display.

You can view a geographic marker in either of the following ways:

- A graphic overlaid on the model space view at the World Coordinate System (WCS) location of the geographic marker.

  NOTE Set the GEOMARKERVISIBILITY system variable to 1 to view the graphic for geographic marker in the drawing.

- A status bar coordinate display that shows the latitude or longitude of a WCS coordinate at the cursor location.

To view the graphic of a geographic marker

1. Enter GEOMARKERVISIBILITY at the Command prompt.
2. Enter 1 as the value.

3 Enter **geographiclocation** at the Command prompt.
Insert a geographic marker using the options available.
A geographic marker is inserted at the coordinates specified.

To view the status bar coordinate display

1 Click Render tab ➤ Sun & Location panel ➤ Set Location. 

2 Insert a geographic marker using the options available.

3 Right-click the status bar display. Click Geographic.
The status bar coordinates display the latitude and longitude values of the geographic location.

Quick Reference

Command

GEOGRAPHICLOCATION

Specifies the geographic location information for a drawing file.

INSERT

Inserts a block or drawing into the current drawing.

XATTACH

Inserts a DWG file as an external reference (xref).

System Variables

GEOLATLONGFORMAT

Controls the format of the latitude or longitude values in the Geographic Location dialog box, and the coordinate status bar in Geographic mode.

GEOMARKERVISIBILITY

Controls the visibility of geographic markers.

Open or Save a Drawing 6

You can use several methods to find and open drawings, even damaged drawings. You can save and backup drawings automatically.

Open a Drawing

You open drawings to work on them just as you do with other Windows applications. In addition, you can choose from several alternative methods.

To open a drawing, you can

- Use Open on the File menu or Quick Access toolbar to display the Select File dialog box. If the FILEDIA system variable is set to 0, the Command prompt version displays instead of a file navigation dialog box.
- Double-click a drawing in Windows Explorer to launch AutoCAD® and open the drawing. If the program is already running, the drawing opens in the current session rather than in a second session.
- Drag a drawing from Windows Explorer into AutoCAD.
  If you drop a drawing anywhere outside the drawing area—for example, the command line or the blank space next to the toolbars— the drawing is opened. However, if you drag a single drawing into the drawing area of an open drawing, the new drawing is not opened but inserted as a block reference.
- Use DesignCenter to open drawings.
- Use the Sheet Set Manager to locate and open the drawings in a sheet set.

Work on Drawings During Loading

You can work on drawings before they are fully open. This is useful when you work on large drawings and you want to begin working immediately. To take advantage of this capability, three conditions are required.

- The drawing must have been saved in paper space.
- The OPENPARTIAL system variable must be set to 1.
- The INDEXCTL system variable must be set to a non-zero value.

When these conditions are met, you can create or modify visible objects, pan or zoom, turn off or freeze layers, and any other operation that does not require displaying objects not visible when the drawing was last saved.

NOTE The Quick View feature will not be fully functional during loading under these conditions.

Resolve Missing References

As you open a drawing, you are notified (messages and task dialog boxes) when a reference cannot be located. From the References - Unresolved Reference Files task dialog box, click Update the Location of the Referenced Files to open the External References palette to make changes to missing external references.

The following table outlines some of the references that might be missing and describes how to handle them.

Missing Reference Types	Description
External references	Missing external references are the result of AutoCAD not being able to resolve the last known location of an xref, raster image, or underlay. To resolve a missing external reference, locate the file and update its location using the External References palette. For information about resolving missing referenced drawing files, see Resolve Missing External References. For information about working with raster images and underlays, see Attach Raster Image Files and Attach Files as Underlays.
Shapes	Missing shape files are often the result of custom shapes being used in a linetype. Browse to the missing linetype file, or place the shape file in the folder with

Missing Reference Types	Description
	the drawing or one of the support paths defined in the Options dialog box. For information about custom shape files in linetypes, see Shapes in Custom Linetypes in the Customization Guide.

You can use eTransmit to avoid missing files when sharing drawings with others outside of your company. For more information about eTransmit, see Package a Set of Files for Internet Transmission.

Work with Large Objects

AutoCAD 2010 supports object size limits greater than those available in previous releases. With increased object size limits you can create larger and more complex models. Using increased object size limits can result in compatibility issues with legacy drawing file formats (AutoCAD 2007 and earlier).

When working with drawings that you might need to exchange with others using AutoCAD 2009 and earlier, set the LARGEOBJECTSUPPORT system variable to 0. Setting LARGEOBJECTSUPPORT to 0 warns you when a drawing contains large objects that cannot be opened by a release of the program prior to AutoCAD 2010.

TrustedDWG™ Drawing Files

DWG, DWT, and DWS files *created* with Autodesk applications and RealDWG™-based applications are trusted by Autodesk. When you open a TrustedDWG file, the following icon displays in the application status bar or the drawing status bar.

If the DWGCHECK system variable is set to On (1), an alert box is displayed if

- The drawing file format is AutoCAD Release 14 or later *and*
- The drawing file was *not* originally created by an Autodesk application or RealDWG-based application.

For more information about TrustedDWG, click the TrustedDWG icon.

Recover Defective Drawing Files

In some circumstances, it is possible that a drawing file becomes defective. This can result from hardware problems or transmission errors. If a drawing file is corrupt, you might be able to recover it. See Repair, Restore, or Recover Drawing Files on page 239.

Change the Default Drawing Folder

Each time you start AutoCAD, the *My Documents* folder is the default path in each standard file selection dialog box. Alternatively, you can configure AutoCAD to always default to a specified path by changing the default drawing folder using the REMEMBERFOLDERS system variable.

See also:

- Overview of DesignCenter on page 88
- Work with Sheets in a Sheet Set on page 479
- Open and Save Drawing Files from the Internet
- Overview of Using Markups for Design Review

To open a drawing

1. Click the Application button, and click Open ➤ Drawing.
2. In the Select File dialog box, select one or more files. Click Open.
   The icons on the left side of the dialog box provide quick access to commonly used files and file locations. To reorder the icons, drag them to a new location. To add, modify, or remove an icon, right-click the icon to display a shortcut menu.

Menu: Click File ➤ Open
Command entry: OPEN

Toolbar: Standard  , Quick Access toolbar

To change the default drawing folder

1. At the Command prompt, enter **rememberfolders**, and then enter **0**.
2. On the Windows desktop, right-click the AutoCAD icon. Click Properties.
3. Click the Shortcut tab.
4. In the Start In box, enter the path of the folder that should be the default when you open or save drawing files.

5 Click OK.

To enable support for large object size limits

1 Click the Application button, and click Options.

2 In the Options dialog box, Open and Save tab, under File Save, clear Maintain Drawing Size Compatibility.

3 Click OK.

Command entry: OPTIONS, LARGEOBJECTSUPPORT

Quick Reference

Commands

CLOSE

Closes the current drawing.

CLOSEALL

Closes all currently open drawings.

ETRANSMIT

Packages a set of files for Internet transmission.

OPEN

Opens an existing drawing file.

OPTIONS

Customizes the program settings.

WHOHAS

Displays ownership information for opened drawing files.

System Variables

DWGCHECK

Checks drawings for potential problems when opening them.

FILEDIA

Suppresses display of file navigation dialog boxes.

INDEXCTL

Controls whether layer and spatial indexes are created and saved in drawing files.

LARGEOBJECTSUPPORT

Controls large object size limit support when you open and save drawings.

OPENPARTIAL

Controls whether a drawing file can be worked on before it is fully open.

REMEMBERFOLDERS

Controls the default path displayed in standard file selection dialog boxes.

ROAMABLEROOTPREFIX

Stores the full path to the root folder where roamable customizable files were installed.

Utilities

No entries

Command Modifiers

No entries

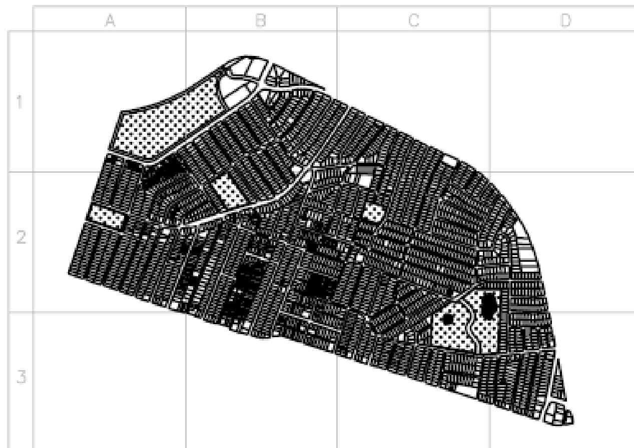
Open Part of a Large Drawing (Partial Load)

If you work with large drawings, you can improve performance by opening only the view and layer geometry that you want to work with.

If you work with large drawings, you can use the Partial Open option of the OPEN command to select which view and layer *geometry* (graphical objects only) that you want to work with in a drawing. For example, if you load geometry from the EXTENTS view and the SITE layer, everything on the SITE layer that falls within the Extents view is loaded into the drawing.

You can only edit what is loaded into the drawing file, but all the drawing's named objects are available in the partially open drawing. Named objects include layers, views, blocks, dimension styles, text styles, viewport configurations, layouts, UCSs, and linetypes.

By organizing large drawings into sectored views, you can load and edit only what you need. For example, if you work with a city plan and need to edit only the southeast sector (sector D3 in the illustration), you can load this drawing area by specifying the predefined view. If you need to edit only the city plot numbers, you can load just the geometry on this specific layer.



After a drawing is partially open, you can load additional geometry from a view, selected area, or layer into the drawing by using PARTIALOAD. The Partial Open option is available only for drawings in AutoCAD 2004 or later format.

To partially open a drawing

1 Click the Application button, and click Open ➤ Drawing.

2 In the Select File dialog box, select a drawing.

3 Click the arrow next to Open. Click Partial Open.

4 In the Partial Open dialog box, select a view; the default view is EXTENTS.

   You can load only geometry from model space views that are saved in the current drawing.

5 Select one or more layers.

   If you do not select a layer or layers to load, no layer geometry is loaded into the drawing but all drawing layers exist in the drawing. If no layer geometry is specified to load into the drawing, no geometry is loaded even if the geometry from a view is specified to load. If you draw objects on a layer that is not loaded, you may be drawing on top of existing geometry that is not loaded in the drawing.

NOTE 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 drawing are not displayed in the Layer Geometry to Load list.

6 If the drawing contains a spatial index, you can select the Use Spatial Index option.

   A spatial index is a list that organizes objects based on their location in space. A spatial index is used to locate the portion of the drawing that is read; this minimizes the time required to open the drawing.

7 If the drawing contains xrefs but you don't want to load them, select Unload All Xrefs on Open.

   NOTE 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.

8 Click Open.

   You can choose to load additional information into the current drawing as long as the drawing is partially open.

Menu: Click File ➤ Open
Command entry: PARTIALOPEN

To load additional geometry into a partially open drawing

1 Click File menu ➤ Partial Load.

   NOTE The Partial Load option is available only if the current drawing is a partially open drawing.

2 In the Partial Load dialog box, select a view, or click Pick Window to define a view.

   The default view is Extents. You can load only geometry from model space views that are saved in the current drawing.

3 Select one or more layers.

   If you do not select a layer or layers to load, no layer geometry is loaded into the drawing, but all drawing layers exist in the drawing. If no layer geometry is specified to load into the drawing, no geometry is loaded even if the geometry from a view is specified to load. If you draw objects on a layer that is not loaded, you may be drawing on top of existing geometry that is not loaded in the drawing. You cannot unload any geometry that is loaded in the current drawing.

4 Click Open.

Command entry: PARTIALOAD

Quick Reference

Commands

PARTIALOAD

Loads additional geometry into a partially opened drawing.

PARTIALOPEN

Loads geometry and named objects from a selected view or layer into a drawing.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

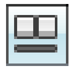
Work with Multiple Open Drawings

You can preview and switch between open drawings and layouts in a drawing and transfer information between open drawings.

Preview Open Drawings and Layouts

With Quick View, you can easily preview and switch between open drawings and the model space and layouts in an open drawing. These are displayed in thumbnail images called Quick View images at the bottom of the application window.

The Quick View tools in the application status bar do the following:

- **Quick View Drawings** 
  Displays all currently open drawings in a row of Quick View drawing images. You can also preview and switch between the model space and layouts in an open drawing when you move your cursor over a Quick View drawing image. For more information, see Switch Between Open Drawings on page 211.

- **Quick View Layouts** 
  Displays the model space and layouts in the current drawing in a row of Quick View layout images. You can right-click a Quick View layout image to view layout options. For more information, see Switch Between Layouts in the Current Drawing on page 217.

The supported file formats are DWG, DWT, DXF, and DWS.

NOTE The application status bar must be displayed to view the Quick View tools.

Quick Reference

Commands

QVDRAWING

Displays open drawings and layouts in a drawing in preview images.

QVDRAWINGCLOSE

Closes preview images of open drawings and layouts in a drawing.

QVLAYOUT

Displays preview images of model space and layouts in a drawing.

QVLAYOUTCLOSE

Closes preview images of model space and layouts in a drawing.

System Variables

STATUSBAR

Controls the display of the application and drawing status bars.

Utilities

No entries

Command Modifiers

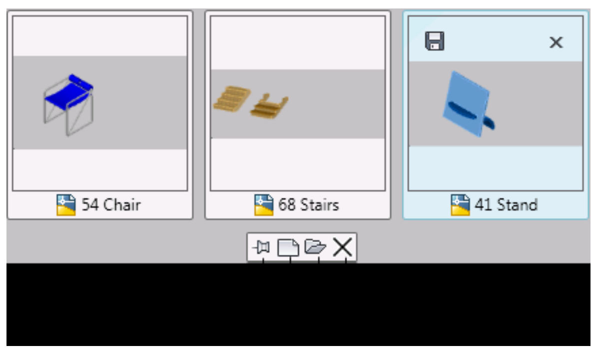
No entries

Switch Between Open Drawings

Preview and switch between all open drawings and layouts in a drawing in two-level structure with the Quick View Drawings tool.

The first level displays the Quick View images of open drawings and the second level displays the images for model space and all layouts in a drawing.

When you click the Quick View Drawings button on the status bar, each open drawing displays as a thumbnail image in a row. The image of the current drawing is highlighted by default. If you move your cursor over an image, all the layouts and the model for that drawing are displayed in a row of images above the Quick View drawing.



The toolbar displayed below the Quick View drawings has the following options:

- **Pin Quick View Drawings**
  Pins the row of Quick View images of drawings so that it is always visible while you are working in the drawing editor.

- **New**
  Creates a drawing that is also displayed at the end of the Quick View images row.

- **Open**

Opens an existing drawing that is also displayed at the end of the Quick View images row.

- **Close**
  Closes all the Quick View images.

You can do any of the following with the Quick View Drawings tool:

- Click a Quick View drawing image to make that drawing current.
- Move your cursor over the Quick View drawing image to display the Save and Close buttons on the top corners of the image to save or close the drawing.
- Move your cursor over a Quick View drawing to display the preview images of the model space and layouts in the drawing.
- Close all drawings, except the drawing you want to work on, from the shortcut menu.
- Close and save all open drawings.
- Manage the display of the drawings in the windows to be tiled vertically or horizontally, or cascaded.

You can access more options for layouts from the Quick View layout image. For more information, see Switch Between Layouts in the Current Drawing on page 217.

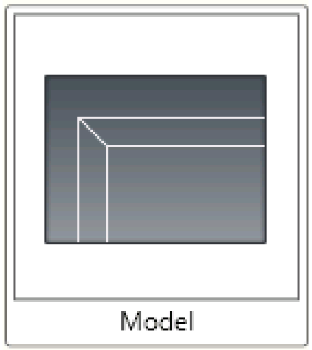
If a row of Quick View images extends beyond the application display area, scroll arrows display on the left or right side of the row. You can scroll to view the other images. You can resize Quick View images dynamically using Ctrl + scrollwheel.

TIP If you often work with many open drawings or many layouts in a drawing, you can use multiple monitors to display complete rows of Quick View images.

Update Quick View Images

Quick View images are not updated dynamically when working on it. They are updated when you switch between the model space and layouts; or use the UPDATETHUMBSNOW command.

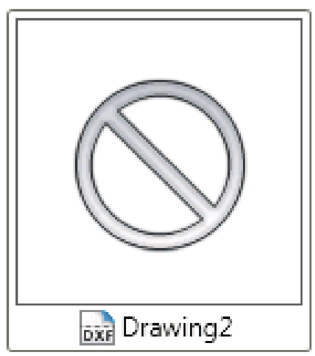
When a drawing does not have a stored image for a model space, a placeholder image is displayed instead.



This placeholder image is displayed in the following cases:

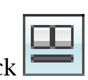
- When the model space is not initialized
- When the UPDATETHUMBNAIL system variable is set to 0. In this case, the image is not updated even if you use the UPDATETHUMBSNOW command.

When a drawing is saved in DXF format, it does not have a stored image for a drawing. The image is not updated even if you use the UPDATETHUMBSNOW command. A static image is displayed instead.



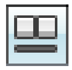
To display Quick View drawings

Do one of the following:

- Click  the status bar
- At the Command prompt, enter **qvdrawing**.

Command entry: QVDRAWING

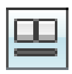
To preview and switch between open drawings

1 Click  the status bar.
A row of Quick View images is displayed at the bottom of the application.

2 Click the Quick View image you want to work on.

Command entry: QVDRAWING

To preview and switch between layouts in a drawing

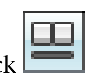
1 Click  the status bar.
A row of Quick View images is displayed at the bottom of the application.

2 Move your cursor over a drawing image to preview the model and layouts in a drawing.

3 Click a Quick View image to make the drawing or layout current.

Command entry: QVDRAWING

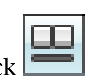
To resize a Quick View image

1 On the status bar, click  .
A row of Quick View images is displayed at the bottom of the application.

2 Press Ctrl + scrollwheel over a Quick View image to resize.

Command entry: QVDRAWING

To view Quick View images not displayed in the application display area

1 On the status bar, click  .
A row of Quick View images is displayed at the bottom of the application.

2 Click the left or right scroll arrow to view Quick View images not shown on the application display area.

Command entry: QVDRAWING

To pin the row of Quick View images

1 On the status bar, click .

A toolbar is displayed below the row of Quick View images.

2 On the toolbar, click the Pin Quick View Drawings button .

The Quick View images are always displayed unless you unpin them. To unpin, click Pin Quick View Drawings button again.

To create a drawing in Quick View

1 On the status bar, click .

A toolbar is displayed below the row of Quick View images.

2 On the toolbar, click the New button .

A new drawing is displayed in a Quick View image.

To open a drawing in Quick View

1 On the status bar, click .

A toolbar is displayed below the row of Quick View images.

2 On the toolbar, click Open .

3 In the Select File dialog box, select one or more files.

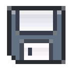
4 Click Open.

To save a drawing in Quick View

1 On the status bar, click .

A row of images is displayed at the bottom of the application.

2 Mouse over the image you want to save.

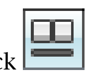
3 Click  displayed on the Quick View image.

To save all open drawings in Quick View

1 On the status bar, click  .
A row of images is displayed at the bottom of the application.

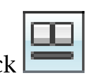
2 Right-click any Quick View drawing image and click Save All.

To close a drawing in Quick View

1 On the status bar, click  .
A row of images is displayed at the bottom of the application.

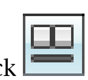
2 Right-click the Quick View drawing image you want to close and click Close.

To close all drawings in Quick View

1 On the status bar, click  .
A row of images is displayed at the bottom of the application.

2 Right-click any Quick View drawing image and click Close All.

To close all drawings except the drawing you want to work on in Quick View

1 On the status bar, click  .
A row of images is displayed at the bottom of the application.

2 Right-click the Quick View image of the drawing you want to work on. Click Close Other Files.

Quick Reference

Commands

QVDRAWING

Displays open drawings and layouts in a drawing in preview images.

QVDRAWINGCLOSE

Closes preview images of open drawings and layouts in a drawing.

SYSWINDOWS

Arranges windows and icons when the application window is shared with external applications.

UPDATETHUMBSNOW

Manually updates thumbnail previews in the Sheet Set Manager and the Quick View tool.

System Variables

QVDRAWINGPIN

Controls the default display state of preview images of drawings.

UPDATETHUMBNAIL

Controls updating of the thumbnail previews in the Sheet Set Manager and Quick View.

Utilities

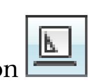
No entries

Command Modifiers

No entries

Switch Between Layouts in the Current Drawing

Preview and switch between the model space and layouts in the current drawing with the Quick View Layouts tool.

When you click the Quick View Layouts button  on the status bar, the model space and layouts in the drawing are displayed in a horizontal row.

You can plot or publish when you move the cursor over a Quick View image of a layout. The toolbar displayed below the Quick View images of drawings has the following options:

- **Pin Quick View Layouts**
  Pins the row of Quick View images of layouts so that it is always visible while you are working in the drawing editor.
- **New Layout**
  Creates a layout that is also displayed as a Quick View image at the end of the row.
- **Publish**
  Launches the Publish dialog box to publish the layouts.
- **Close Quick View Layouts**
  Closes all the Quick View layout images.

You can do any of the following with a Quick View layout image:

- Click the image to display the associated layout or model in the drawing area
- Move your cursor over the image to display the Plot and Publish buttons
- Right-click the image to display a shortcut menu with additional options
- Use Ctrl + scroll wheel to resize Quick View images dynamically

TIP If you often work with many open drawings or many layouts in a drawing, you can use multiple monitors to display complete rows of Quick View images.

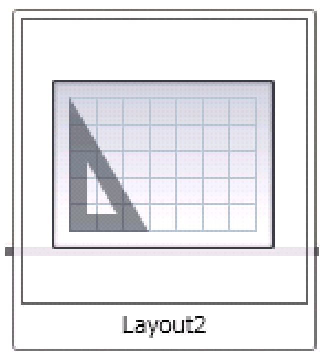
If a row of Quick View images extends beyond the application display area, scroll arrows display on the left or right side of the row. You can scroll to view the other images.

NOTE The Quick View feature will not be fully functional if the drawing is still opening and not fully loaded.

Update Quick View Layout Images

Quick View images for layouts are not updated dynamically when working on a layout. They are updated when you switch between layouts or use the UPDATETHUMBSNOW command.

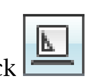
When a drawing does not have a stored image for a layout, a placeholder image is displayed instead.



This placeholder image is displayed in the following cases:

- When the layout is not initialized
- When the UPDATETHUMBNAIL system variable is set to 0. In this case, the image is not updated even if you use the UPDATETHUMBSNOW command.

To preview model space and layouts in a drawing

- On the status bar, click  .

A row of Quick View images of model space and layouts in a drawing is displayed at the bottom of the application.

Command entry: QVLAYOUT

To update a thumbnail image

NOTE Verify that **updatethumbnail** is not set to **0**

---

1 At the Command prompt, enter **updatethumbsnow**.

This command ensures that thumbnail images are updated.

2 On the status bar, click  .

A row of Quick View images of model space and layouts in a drawing displays at the bottom of the application.

To make a model space or layout current

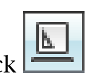
1 On the status bar, click  .

A row of Quick View images of model space and layouts in a drawing is displayed at the bottom of the application.

2 Click a model space or layout image to make it current.

Command entry: QVLAYOUT

To specify the page setup for a layout in Quick View

1 On the status bar, click  .

A row of Quick View images of model space and layouts in a drawing is displayed at the bottom of the application.

2 Right-click the Quick View image. Click Page Setup.

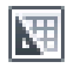
Command entry: QVLAYOUT

To create a layout in Quick View

1 On the status bar, click .

A row of Quick View images of model space and layouts in a drawing is displayed at the bottom of the application. A toolbar is displayed below the Quick View images.

2 Do one of the following:

- Right-click the Quick View image. Click New Layout.
- On the toolbar, click .

A Quick View image of a new layout is displayed.

Command entry: QVLAYOUT

To rename a layout in Quick View

1 On the status bar, click .

A row of Quick View images of model space and layouts in a drawing is displayed at the bottom of the application.

2 Double-click the layout name you want to rename.

3 Enter a new name for the layout.

Command entry: QVLAYOUT

To import a layout from a template in Quick View

1 On the status bar, click .

A row of Quick View images of model space and layouts in a drawing is displayed at the bottom of the application.

2 Right-click a Quick View image. Click From Template.

3 In the Select File dialog box, select a DWT or DWG file to import a layout from. Click Open. In the Insert Layout(s) dialog box, select a layout to import.

4 Enter a new name for the layout. Click OK.

Command entry: QVLAYOUT

To arrange layouts in Quick View

1 On the status bar, click  .

A row of Quick View images of model space and layouts in a drawing is displayed at the bottom of the application.

2 Right-click the Quick View image you want to move. Click Move or Copy.

3 In the Move or Copy dialog box, select a new position for the model or layout image that should be placed immediately after the moved or copied layout. If you want to move the layout tab to the end of the list of layout tabs, select Move to End.

4 Click OK.

Command entry: QVLAYOUT

To duplicate a layout in Quick View

1 On the status bar, click  .

A row of Quick View images of model space and layouts in a drawing is displayed at the bottom of the application.

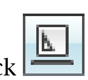
2 Right-click the Quick View image you want to move. Click Move or Copy.

3 In the Move or Copy dialog box, select a new position for the new layout image. Click OK.

NOTE This option is not available in model space.

Command entry: QVLAYOUT

To arrange layouts in Quick View

1 On the status bar, click  .

A row of Quick View images of model space and layouts in a drawing is displayed at the bottom of the application.

2 Right-click the Quick View image you want to move. Click Move or Copy.

3 In the Move or Copy dialog box, select a new position for the model or layout image that should be placed immediately after the moved or copied layout. If you want to move the layout tab to the end of the list of layout tabs, select Move to End.

4 Click OK.

Command entry: QVLAYOUT

To activate the previous layout in Quick View

1 On the status bar, click .

A row of Quick View images of model space and layouts in a drawing is displayed at the bottom of the application.

2 Right-click a Quick View image. Click Activate Previous Layout.

Command entry: QVLAYOUT

To select all layouts in Quick View

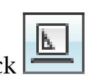
1 On the status bar, click .

A row of Quick View images of model space and layouts in a drawing is displayed at the bottom of the application.

2 Right-click a Quick View image. Click Select All Layouts.

Command entry: QVLAYOUT

To export a layout to model space in Quick View

1 On the status bar, click .

A row of Quick View images of model space and layouts in a drawing is displayed at the bottom of the application.

2 Right-click a Quick View image. Click Export Layout to a Model.

3 In the Export Layout to Model Space Drawing dialog box, enter a name for the new file. Click Save.

Command entry: QVLAYOUT

To plot a layout in Quick View

1 On the status bar, click  .

A row of Quick View images of model space and layouts in a drawing is displayed at the bottom of the application.

2 On the top-left corner of a layout image, click  .

3 In the Plot dialog box, select the plot settings. Click OK.

Command entry: QVLAYOUT

To publish a layout in Quick View

1 On the status bar, click  .

A row of Quick View images of model space and layouts in a drawing is displayed at the bottom of the application.

2 On the top-left corner of a layout image, click  .

3 In the Publish dialog box, select the publish settings. Click OK.

Command entry: QVLAYOUT

Quick Reference

Commands

PLOT

Plots a drawing to a plotter, printer, or file.

PUBLISH

Publishes drawings to DWF, DWFx, and PDF files, or to plotters.

QVLAYOUT

Displays preview images of model space and layouts in a drawing.

QVLAYOUTCLOSE

Closes preview images of model space and layouts in a drawing.

UPDATETHUMBSNOW

Manually updates thumbnail previews in the Sheet Set Manager and the Quick View tool.

System Variables

QVLAYOUTPIN

Controls the default display state of preview images of model space and layouts in a drawing.

UPDATETHUMBNAIL

Controls updating of the thumbnail previews in the Sheet Set Manager and Quick View.

Transfer Information between Open Drawings

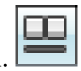
You can easily transfer information between drawings that are open in a single session.

When you open multiple drawings in a single session, you can

- Reference other drawings.
- Copy and paste between drawings.
- Drag selected objects from one drawing to another with the right-click button on your pointing device.
- Use Match Properties (MATCHPROP) to copy properties from objects in one drawing to objects in another drawing.
- Use object snaps, the Copy with Basepoint (COPYBASE) command, and the Paste to Original Coordinates (PASTEORIG) command to ensure accurate placement.

To switch between open drawings

Do one of the following to switch between open drawings:

- On the status bar, click the Quick View Drawings tool. 
- Click anywhere in a drawing to make it active.
- Use Ctrl+F6 or Ctrl+Tab.

Command entry: TASKBAR

To display multiple open drawings on the Windows taskbar

1. At the Command prompt, enter taskbar, and then enter 1.
2. To switch between drawings, press Alt+Tab.

Command entry: TASKBAR

To display only the active drawing on the Windows taskbar

- At the Command prompt, enter **taskbar**, and then enter 0.

Command entry: TASKBAR

Quick Reference

Commands

OPTIONS

Customizes the program settings.

TASKBAR

Controls how drawings are displayed on the Windows taskbar.

Save a Drawing

You save drawing files for later use just as you do with other Microsoft Windows applications. You can also set up automatic saving and backup files and save only selected objects.

When you work on a drawing, you should save it frequently. Saving protects you from losing work in the event of a power failure or other unexpected event. If you want to create a new version of a drawing without affecting the original drawing, you can save it under another name.

The file extension for drawing files is *.dwg*, and unless you change the default file format in which drawings are saved, drawings are saved in the latest drawing file format. This format is optimized for file compression and for use on a network.

The character limit for a DWG file name (including its path) is 256 characters.

NOTE If the FILEDIA system variable is set to 0, the Command prompt version displays instead of a file navigation dialog box.

Save Part of a Drawing File

If you want to create a new drawing file from part of an existing drawing, you use the WBLOCK command. With the command, you can select objects or specify a block definition in your current drawing and save them to a new drawing file. You can also save a description with the new drawing.

Save to a Different Type of Drawing File

You can save a drawing to an earlier version of the drawing format (DWG) or drawing interchange format (DXF), or save a drawing as a template file. Choose the format from Files of Type in the Save Drawing As dialog box.

Save with Visual Fidelity for Annotative Objects

When working with annotative on page 1923 objects, this option allows you to maintain visual fidelity for these objects when they are viewed in AutoCAD 2007 and earlier releases. Visual fidelity is controlled by the SAVEFIDELITY system variable.

If you work primarily in model space, it is recommended that you turn off visual fidelity (set SAVEFIDELITY to 0). However, if you need to exchange drawings with other users, and layout fidelity is most important, then visual fidelity should be turned on (set SAVEFIDELITY to 1).

NOTE The SAVEFIDELITY system variable does not effect saving a drawing to the AutoCAD 2010 drawing or DXF file formats.

Annotative objects may have multiple scale representations on page 1950. When visual fidelity is on, annotative objects are decomposed and scale representations are saved (in an anonymous block on page 1923) to separate layers, which are named based on their original layer and appended with a number. If you explode the block in AutoCAD 2007 or earlier releases, and then open the drawing in AutoCAD 2008 or later releases, each scale representation becomes a separate annotative object, each with one annotation scale. It is not recommended that you edit or create objects on these layers when working with a drawing created in AutoCAD 2008 and later releases in AutoCAD 2007 and earlier releases.

When this option is not selected, a single model space representation is displayed on the Model tab. More annotation objects may be displayed on the Model tab depending on the ANNOALLVISIBLE setting. Also, more objects may be displayed in paper space viewports at different sizes than in AutoCAD 2008 and later releases.

Reduce the Time Required to Save a Drawing File

You can reduce the time required to save a drawing file if you specify incremental saves rather than full saves. An incremental save updates only those portions of the saved drawing file that you changed.

When you use incremental saves, drawing files will contain a percentage of potentially wasted space. This percentage increases after each incremental save until it reaches a specified maximum, at which time a full save is performed instead. You can set the incremental save percentage in the Open and Save tab of the Options dialog box or by setting the value of the system variable ISAVEPERCENT. If you set the value of ISAVEPERCENT to 0, all saves are full saves.

To reduce the size of drawing files, it is recommended that you perform a full save (with ISAVEPERCENT set to 0) before transmitting or archiving a drawing.

Work Internationally

If you share drawing files with companies in other countries and regions, the drawing file names might contain characters that are not used in other languages.

If a drawing file is created in a different language version of Windows, the following will occur:

- If support for the language is installed, the file name characters are visible in Windows Explorer.
- If support for the language is *not* installed, the file name characters appear as a series of boxes in Windows Explorer and a dialog box is displayed offering to install the language pack.

In *either* case, you will be able to open the drawing file beginning with AutoCAD 2007 or AutoCAD LT 2007 because these products are Unicode-compliant applications.

NOTE If you share drawing files with companies using earlier releases of the product, you can avoid file name issues for Asian languages and languages that use accented characters. In those circumstances, do not use high ASCII values, or values of 80 hexadecimal and above, when creating a file name.

Maintain Compatibility with Large Object Limits

Drawings saved to a legacy drawing file format (AutoCAD 2007 or earlier) do not support objects greater than 256MB. With the AutoCAD 2010 drawing file format, these limitations have been removed allowing you to save objects that are greater in size.

When saving to a legacy drawing file format (AutoCAD 2007 or earlier), the drawing cannot contain large objects; there might be compatibility issues with trying to open the drawing. The LARGEOBJECTSUPPORT system variable controls the large object size limits used and the warning messages displayed when a drawing is saved.

The following explains how object size limits for drawings is determined:

- Drawing files cannot exceed an internal size limit of 4GB. This size is based on the total size of all objects in a drawing when uncompressed. Since a drawing file is normally compressed, the final size of a saved drawing file on disk will vary based on the size and number of objects in a drawing.
- Each individual object in a drawing cannot exceed an uncompressed size limit of 256MB. For example, a mesh object, when saved to a file and compressed, might be 75MB in size while the same object when uncompressed might be 257MB.

In these situations, the drawing cannot be saved to an AutoCAD 2007 or earlier file format until the issues are resolved. You can resolve the size limits by breaking the drawing or objects up into several drawings or objects.

NOTE When working with the 64-bit release of AutoCAD, you can work more efficiently with large objects and drawings. However, the drawing files you create might be too large to open with the 32-bit release of AutoCAD.

See also:

- Save Drawings to Previous Drawing File Formats

- Work with Drawings in Earlier Releases
- Export Drawings to Other File Formats
- Create Drawing Files for Use as Blocks on page 858
- Add Identifying Information to Drawings on page 188
- Create and Restore Backup Files on page 243
- Share Drawing Files Internationally
- Open and Save Drawing Files from the Internet

To save a drawing

1. Click File menu ➤ Save.

   If you previously saved and named the drawing, any changes made are saved and the Command prompt is redisplayed. If you have never saved the drawing, the Save Drawing As dialog box is displayed.

2. In the Save Drawing As dialog box, under File Name, enter the new drawing name; the file extension is not required. Click Save.

Command entry: SAVE

Toolbar: Standard 

To save a drawing automatically

1. Click the Application button. At the bottom of the Application menu, click the Options button.
2. In the Options dialog box, Open and Save tab, select Automatic Save.
3. Enter a number in Minutes Between Saves. Click OK.

Command entry: OPTIONS

To save a backup of the previous version every time the drawing is saved

1. Click the Application button. At the bottom of the Application menu, click the Options button.

2 In the Options dialog box, Open and Save tab, select Create Backup Copy with Each Save. Click OK.

Command entry: OPTIONS

To save selected objects into a new drawing file

1 At the Command prompt, enter **wblock**.

2 In the Write Block dialog box, under Objects, select Objects.

3 Under Base Point, click Pick Point.

4 Specify the base point in the drawing area, or enter coordinate values in the *X*, *Y*, and *Z* boxes.

5 Under Objects, click the Select Objects button.

6 Select the objects in the drawing area.

7 Under Objects, select an option to indicate whether you want to retain, convert, or delete the selected objects.

8 In the File Name box, enter the name for the new drawing file.

9 In the Location box, specify the folder for the new drawing file.

10 In the Insert Units box, you can select a different base unit. This option provides a method to scale objects automatically in the new drawing file.

11 Click OK.

The objects you selected in the drawing are saved in a new drawing file.

Command entry: WBLOCK

To save a block definition as a new drawing file

1 At the Command prompt, enter **wblock**.

2 In the Write Block dialog box, select Block.

3 Click in the box next to Block. Select the block definition you want to save.

4 In the File Name box, enter the name for the new drawing file.

5 In the Location box, specify the folder for the new drawing file.

6 In the Insert Units box, you can select a different base unit. This option provides a method to scale objects automatically in the new drawing file.

7 Click OK.

   The objects in the specified block definition are saved as objects in a new drawing file. The insertion base point of the block definition will be located at the origin point (0,0,0) in the new drawing.

Command entry: WBLOCK

To maintain visual fidelity for annotative objects

1 Click the Application button, and click Options.

2 In the Options dialog box, Open and Save tab, under File Save, select Maintain Visual Fidelity for Annotative Objects.

3 Click OK.

Command entry: OPTIONS, SAVEFIDELITY

To maintain large object compatibility with legacy drawing file formats

1 Click the Application button, click Options.

2 In the Options dialog box, Open and Save tab, under File Save, select Maintain Drawing Size Compatibility.

3 Click OK.

Command entry: OPTIONS, LARGEOBJECTSUPPORT

Quick Reference

Commands

BLOCK

Creates a block definition from selected objects.

OPTIONS

Customizes the program settings.

QSAVE

Saves the current drawing using the file format specified in the Options dialog box.

QUIT

Exits the program.

SAVE

Saves the drawing under the current file name or a specified name.

SAVEAS

Saves a copy of the current drawing under a new file name.

WBLOCK

Writes objects or a block to a new drawing file.

System Variables

DWGCHECK

Checks drawings for potential problems when opening them.

DWGNAME

Stores the name of the current drawing.

DWGPREFIX

Stores the drive and folder prefix for the drawing.

DWGTITLED

Indicates whether the current drawing has been named.

FILEDIA

Suppresses display of file navigation dialog boxes.

ISAVEBAK

Improves the speed of incremental saves, especially for large drawings.

ISAVEPERCENT

Determines the amount of wasted space tolerated in a drawing file.

LARGEOBJECTSUPPORT

Controls large object size limit support when you open and save drawings.

RASTERPREVIEW

Controls whether BMP preview images are saved with the drawing.

SAVEFIDELITY

Controls whether the drawing is saved with visual fidelity.

SAVEFILE

Stores the current automatic save file name.

SAVEFILEPATH

Specifies the path to the directory for all automatic save files for the current session.

SAVENAME

Displays the file name and directory path of the most recently saved drawing.

SAVETIME

Sets the automatic save interval, in minutes.

Find a Drawing File

You can search for a drawing using name, location, and date filters, properties such as keywords that you added to the drawing, or text strings containing a specific word or phrase.

- Use the Search tool in Microsoft® Windows® to search for drawings using name, location, and date filters. You can also specify a word or phrase contained in the drawing file. You can search for all textual data except text in tables and fields, and xrefs within drawing files. Supported drawing file types include DWG, DWF, DWT, and DWS.
- Use the Search dialog box for DesignCenter™ to search for Microsoft Windows file properties, such as title or keyword, that you added to drawings.
- Use the Select File dialog box for the OPEN command to display drawing file previews. When the RASTERPREVIEW system variable is on, a raster preview image is automatically generated and stored with the drawing when you save it.

See also:

- Add Identifying Information to Drawings on page 188

- Access Content with DesignCenter on page 92
- Work with Sheets in a Sheet Set on page 479

To search for files

1 Click the Application button, and click Open ➤ Drawing.

2 In the Select File dialog box, click Tools ➤ Find

3 In the Find dialog box, Name & Location tab, specify a file type, file name, and path.

You can use wild cards when specifying a file name.

4 On the Date Modified tab, click All Files, or click Find All Files Created or Modified to specify a date filter.

You can search for drawings modified between a specified range of dates or within a specified number of months or days.

5 Click Find Now.

6 Select one or more files from the search results. Click OK.

7 In the Select File dialog box, click Open.

Command entry: OPEN
Menu: File ➤ Open

Toolbar: Standard 

Quick Reference

Commands

OPEN

Opens an existing drawing file.

Specify Search Paths and File Locations

You can set the search path to drawing support files such as text fonts, drawings, linetypes, and hatch patterns. You also can specify the location of temporary files, which is important when working in a network environment.

The Files tab of the Options dialog box is where you set the search path that is used by the program to find drawing support files such as text fonts, drawings, linetypes, and hatch patterns. The MYDOCUMENTSPREFIX system variable stores the location of the *My Documents* folder for the current user.

The working search path for drawing support files lists paths that are valid and exist in the current system folder structure (including system network mapping). Using these options helps improve performance when these files are loaded.

Using the Files tab of the Options dialog box, you can also specify the location of temporary files. Temporary files are created on disk, and then deleted when you exit the program. The temporary folder is set to the location that Microsoft Windows uses. If you plan to run this program from a write-protected folder (for example, if you work on a network or open files on a CD), specify a different location for your temporary files.

The temporary folder that you specify must not be write-protected, and the drive containing the folder should have sufficient disk space for the temporary files. It is recommended that you manually delete the files from this folder on a regular basis to ensure sufficient space is provided for temporary files. If not enough space is available for temporary files, you may experience errors or instability in the program.

If you want to use a file that contains custom interface elements, specify it in the Customizations Files item on the Files tab of the Options dialog box. The default customization file is *acad.cuix*.

To change a search path

1. Click the Application button. At the bottom of the Application menu, click the Options button.
2. In the Options dialog box, Files tab, click the plus sign (+) to the left of the kind of path you want to change.
3. Select the path you want to change.
4. Click Browse, and then search drives and folders until you find the one you want.
5. Select the drive and folder that you want to use. Click OK.

Command entry: OPTIONS

Quick Reference

Commands

OPTIONS

Customizes the program settings.

System Variables

MYDOCUMENTSPREFIX

Stores the full path to the My Documents folder for the user currently logged on.

Repair, Restore, or Recover Drawing Files

7

If a drawing file is damaged or if your program terminates unexpectedly, you can recover some or all of the data by using commands to find and correct errors, or by reverting to a backup file.

Repair a Damaged Drawing File

If a drawing file is damaged, you can recover some or all of the data by using commands to find and correct errors.

Repair and Recovery

When an error occurs, diagnostic information is recorded in the *acad.err* file, which you can use to report a problem.

A drawing file is marked as damaged if corrupted data is detected, or if you request that the drawing be saved after a program failure. If the damage is minor, sometimes you can repair the drawing simply by opening it. Otherwise, you can use the following:

- RECOVER. Performs an audit on, and attempts to open, any drawing file.
- RECOVERALL. Similar to recover, it additionally operates on all nested xrefs. The results are displayed in the Drawing Recovery Log window.
- AUDIT. Finds and corrects errors in the currently open drawing file.

Example: Auditing Files

Auditing a file generates a description of problems with a drawing file and recommendations for correcting them. As you start the audit, you specify whether you want the program to try to fix the problems it encounters. The report is similar to the following example:

Auditing Header
DXF NAME Current Value Validation Default
PDMODE 990 - 2040
UCSFOLLOW 811 or 0
Error found in auditing header variables
4 Blocks audited
Pass 1 4 objects audited
Pass 2 4 objects audited
Total errors found 2 fixed 2

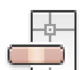
If you chose not to correct the errors, the last statement changes to

Total errors found 2 fixed 0.

The output from a recovery audit is written to an audit log (ADT) file if the AUDITCTL system variable is set to 1 (On).

Recovery does not necessarily preserve the high-level consistency of the drawing file. The program extracts as much material as it can from the damaged file.

To repair a damaged drawing file

1. Click the Application button, and click Drawing Utilities ➤ Recover ➤ Recover. 
2. In the Select File dialog box, select a file. Click Open.

   After the audit, all objects with errors are placed in the Previous selection set for easy access. The output from the audit is written to an audit log (ADT) file if the AUDITCTL system variable is set to 1 (On).

Command entry: RECOVER, RECOVERALL

To repair a damaged drawing file due to a system failure

1. If the program encounters a problem and cannot continue, it displays an error message and, for some errors, an error code. Record the error code number, save the changes if possible, and exit to the operating system.
2. Restart the program.

3 In the Drawing Recovery window, under Backup Files, double-click the drawing node to expand it. On the list, double-click one of the drawing or backup files to open it.

   If the program detects that the drawing has been damaged, a message is displayed asking if you want to proceed.

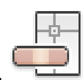
4 Enter **y** to proceed.

   As the program attempts to repair the drawing, a diagnostic report is displayed. The output from the audit is written to an audit log (ADT) file if the AUDITCTL system variable is set to 1 (On).

5 Depending on whether the repair is successful, do one of the following:
   - If the repair is successful, the drawing opens. Save the drawing file.
   - If the program cannot repair the file, a message is displayed. In that case, choose one of the other drawing or backup files listed in the Drawing Recovery window beginning with step 3.

Command entry: RECOVER

To repair an open drawing

1 Click the Application button, and click Drawing ➤ Audit. 

2 At the `Fix Any Errors Detected?` prompt, enter **y** or **n**.

   AUDIT places all objects with errors in the Previous selection set for easy access. The output from the audit is written to an audit log (ADT) file if the AUDITCTL system variable is set to 1 (On).

   NOTE If the drawing contains errors that AUDIT cannot fix, try using RECOVER. RECOVER repairs any specified DWG file that is not the current drawing file.

Command entry: AUDIT

To restore a drawing from a backup file

1 In Windows Explorer, locate the backup file identified by the *.bak* file extension.

2 Right-click the backup file. Click Rename.

3 Type a new name using the *.dwg* file extension.

4 Open the file as you would open any other drawing file.

Quick Reference

Commands

AUDIT

Evaluates the integrity of a drawing and corrects some errors.

DRAWINGRECOVERY

Displays a list of drawing files that can be recovered after a program or system failure.

OPTIONS

Customizes the program settings.

RECOVER

Repairs and then opens a damaged drawing file.

RECOVERALL

Repairs and then opens a damaged drawing file.

System Variables

AUDITCTL

Controls whether AUDIT creates an audit report (ADT) file.

REPORTERROR

Controls whether an error report can be sent to Autodesk if the program closes unexpectedly.

Utilities

No entries

Command Modifiers

No entries

Create and Restore Backup Files

Backup files help ensure the safety of your drawing data. If a problem occurs, you can restore a drawing backup file.

Computer hardware problems, power failures or surges, user mistakes, or software problems can cause errors in a drawing. By saving your work frequently, you can ensure a minimum of lost data if your system fails for any reason. If a problem occurs, you can restore a drawing backup file.

Use Backup Files

In the Options dialog box, on the Open and Save tab, you can specify that backup files are created when you save drawings. If you do, each time you save a drawing, the previous version of your drawing is saved to a file with the same name and a *.bak* file extension. The backup file is located in the same folder as the drawing file.

You can revert to your backup version by renaming the *.bak* file in Windows Explorer to a file with a *.dwg* extension. You may want to copy it to a different folder to avoid overwriting your original file.

Save Your Drawing Automatically at Specified Intervals

If you turn the automatic save option on, your drawing is saved at specified time intervals. By default, files saved automatically are temporarily assigned the name *filename_a_b_nnnn.sv$*.

- *Filename* is the current drawing name.
- *a* is the number of open instances of the same drawing file in the same work session.
- *b* is the number of open instances of the same drawing in different work sessions.
- *nnnn* is a random number.

These temporary files are automatically deleted when a drawing closes normally. In the event of a program failure or a power failure, these files are not deleted.

To recover a previous version of your drawing from the automatically saved file, rename the file using a *.dwg* extension in place of the *.sv$* extension before you close the program.

See also:

- Recover from a System Failure on page 245

To restore a drawing from a backup file

1. In Windows Explorer, locate the backup file identified by the *.bak* file extension.
2. Select the file you want to rename. You do not need to open it.
3. Click File menu ➤ Rename
4. Enter a new name using the *.dwg* file extension.
5. Open the file as you would open any other drawing file.

Quick Reference

Commands

OPTIONS

Customizes the program settings.

System Variables

ISAVEBAK

Improves the speed of incremental saves, especially for large drawings.

Utilities

No entries

Command Modifiers

No entries

Recover from a System Failure

A hardware problem, power failure, or software problem can cause this program to terminate unexpectedly. If this happens, you can restore the drawing files that were open.

If the program fails, you can save your current work to a different file. This file uses the format, *DrawingFileName_recover.dwg*, where *DrawingFileName* is the file name of your current drawing.

Resolve Drawing Files

After a program or system failure, the Drawing Recovery Manager opens the next time you start AutoCAD. Drawing Recovery Manager displays a list of all drawing files that were open, including the following drawing file types:

- Drawing files (DWG)
- Drawing template files (DWT)
- Drawing Standards files (DWS)

NOTE Unsaved drawings that are open at the time of an unexpected failure are not tracked by the Drawing Recovery Manager. Be sure to save your work after you begin, and regularly thereafter.

For each drawing, you can open and choose from the following files if they exist:

- *DrawingFileName*_recover.dwg
- *DrawingFileName_a_b_nnnn*.sv$
- *DrawingFileName*.dwg
- *DrawingFileName*.bak

NOTE The drawing, backup, and recover files are listed in the order of their time stamps—the time when they were last saved.

Double-click a top-level drawing node listed under Backup Files to display up to four files as listed above. Right-click any node under Backup Files to display shortcut menu options.

If you close the Drawing Recovery window before resolving all affected drawings, you can open Drawing Recovery at a later time with the DRAWINGRECOVERY command.

Send an Error Report Automatically to Autodesk

If the program encounters a problem and closes unexpectedly, you can send an error report to help Autodesk diagnose problems with the software. The error report includes information about the state of your system at the time the error occurred. You can also add other information, such as what you were doing at the time of the error. The REPORTERROR system variable controls whether the error-reporting feature is available.

To open the Drawing Recovery Manager

- Click the Application button, and click Drawing Utilities ➤ Open the Drawing Recovery Manager.
  Any drawings that need to be restored from an unexpected program or system failure are listed under Backup Files.

Command entry: DRAWINGRECOVERY

To restore a drawing using the Drawing Recovery Manager

1. If necessary, click the Application button, and click Drawing Utilities ➤ Open the Drawing Recovery Manager.
2. In the Drawing Recovery Manager, under Backup Files, double-click a drawing node to list all available drawing and backup files.
3. Double-click a file to open it.

   If the drawing file is damaged, the drawing is automatically repaired, if possible.

Command entry: DRAWINGRECOVERY

To remove a drawing from the Drawing Recovery Manager

1 If necessary, click the Application button, and click Drawing Utilities ➤ Open the Drawing Recovery Manager.

2 Do *one* of the following:
   - Restore the drawing and then save it.
   - Right-click a drawing node. Click Remove.

Command entry: DRAWINGRECOVERY

To expand or collapse all nodes in the Drawing Recovery Manager

- Under the last drawing node listed, right-click in the Backup Files area. Click Collapse All.

Command entry: DRAWINGRECOVERY

To turn error reporting on or off

1 At the command prompt, enter **reporterror**.

2 Enter **0** to turn off error reporting, or enter **1** to turn on error reporting.

Quick Reference

Commands

DRAWINGRECOVERY

Displays a list of drawing files that can be recovered after a program or system failure.

DRAWINGRECOVERYHIDE

Closes the Drawing Recovery Manager.

OPTIONS

Customizes the program settings.

RECOVER

Repairs and then opens a damaged drawing file.

System Variables

DRSTATE

Indicates whether the Drawing Recovery Manager window is open or closed.

ISAVEBAK

Improves the speed of incremental saves, especially for large drawings.

RECOVERYMODE

Controls whether drawing recovery information is recorded after a system failure.

REPORTERROR

Controls whether an error report can be sent to Autodesk if the program closes unexpectedly.

Utilities

No entries

Command Modifiers

No entries

Maintain Standards in Drawings

8

Drawings are easier to interpret if you set standards to enforce consistency. You can set standards for layer names, dimension styles, and other elements; check drawings against these standards; and then change any properties that do not conform.

Overview of CAD Standards

You can create a standards file to define common properties in order to maintain consistency throughout your drawing files. Standards define a set of common properties for named objects such as layers and text styles. You or your CAD manager can create, apply, and audit standards in drawings to enforce consistency. Because standards make it easier for others to interpret drawings, standards are particularly useful in collaborative environments, where many individuals contribute to the creation of a drawing.

Named Objects for Standards-Checking

You can create standards for the following named objects:

- Layers
- Text styles
- Linetypes
- Dimension styles

Standards File

After you define standards, you save them as a standards file. You can then associate the standards file with one or more drawing files. After you associate a standards file with a drawing, you should periodically check the drawing to make sure it conforms with the standards.

How a Standards Audit Works

When you check a drawing for standards violations, each named object of a specific type is checked against the standards files associated with the drawing. For example, each layer in the drawing is checked against the layers in the standards file.

A standards audit can uncover two types of problems:

- An object with a nonstandard name is present in the drawing being checked. For example, a layer named WALL is present in the drawing but not in any associated standards files.
- A named object in a drawing matches the name of one in a standards file, but their properties are different. For example, in the drawing the WALL layer is yellow, but the standards file specifies red for the WALL layer.

When you fix objects with nonstandard names, the nonstandard objects are purged from the drawing. Any drawing objects that are associated with the nonstandard object are transferred to a replacement standard object that you specify. For example, you fix a nonstandard layer, WALL, and replace it with the standard ARCH-WALL. In this example, choosing Fix in the Check Standards dialog box transfers all objects from layer WALL to layer ARCH-WALL and then purges layer WALL from the drawing.

Standards Plug-Ins

The auditing process uses standards *plug-ins,* applications that define the rules for the properties that are checked for individual named objects. Layers, dimension styles, linetypes, and text styles are each checked against their corresponding plug-ins. You can specify which plug-ins to use when checking a drawing for standards violations. Autodesk or third-party developers may add standards plug-ins for checking additional drawing properties.

All plug-ins check all properties for each named object except for the layer plug-in. The following layer properties are checked when using the layer plug-in:

- Color

- Linetype
- Lineweight
- Plot style mode
- Plot style name (when the PSTYLEMODE system variable is set to 0)

The following layer properties are *not* checked by the layer plug-in:

- On/Off
- Freeze/Thaw
- Lock
- Plot/No Plot

Standards Settings

A variety of settings that may be useful to CAD managers are available in the CAD Standards Settings dialog box. This dialog box can be accessed by clicking Settings in the Check Standards and Configure Standards dialog boxes.

Quick Reference

Commands

CHECKSTANDARDS

Checks the current drawing for standards violations.

STANDARDS

Manages the association of standards files with drawings.

System Variables

STANDARDSVIOLATION

Specifies whether a user is notified of standards violations in the current drawing when a nonstandard object is created or modified.

Utilities

Batch Standards Checker

Audits a set of drawings for standards violations.

Command Modifiers

No entries

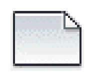
Define Standards

To set standards, you create a file that defines properties for layers, dimension styles, linetypes, and text styles, and you save it as a standards file with the *.dws* file name extension.

Depending on how you organize your projects, you may decide to create and associate more than one project-specific standards file with an individual drawing. When you audit the drawing file, conflicts may arise between settings in the standards files. For example, one standards file specifies that the layer WALL is yellow, and another standards file specifies that it is red. In the case of conflicts, the first standards file associated with the drawing takes precedence. If necessary, you can change the order of the standards files to change the precedence.

If you want to audit drawings using just a specific plug-in, you can specify the plug-in when defining your standards file. For example, if recent changes to a drawing are limited to text changes, you may want to audit the drawing using only the layers and text styles plug-ins to save time. By default, all plug-ins are used when auditing drawings for standards violations.

To create a standards file

1. Click the Application button, and click New menu ➤ Drawing. 
2. Enter a template file name, or press Enter to continue.
3. In a new drawing, create any layers, dimension styles, linetypes, and text styles that you want to be part of the standards file.
4. Click the Application button, and click Save As ➤ AutoCAD Drawing Standards.

   DWS files must be saved in the current drawing file format. To create a DWS file in a previous drawing file format, save the file in the desired DWG format, and then rename the DWG file using a *.dws* extension.
5. In File Name, enter a name for the standards file. Click Save.

To associate a standards file with the current drawing

1 Click Manage tab ➤ Cad Standards panel ➤ Configure.

2 In the Configure Standards dialog box, Standards tab, click the plus (+) button (Add Standards File).

3 In the Select Standards File dialog box, select a standards file. Click Open.

4 (Optional) Repeat steps 2 and 3 if you want to associate additional standards files with the current drawing.

5 Click OK.

Command entry: STANDARDS

Toolbar: CAD Standards

To remove a standards file from the current drawing

1 Click Manage tab ➤ Cad Standards panel ➤ Configure.

2 In the Configure Standards dialog box, Standards tab, select a standards file in Standards Files Associated with the Current Drawing.

3 Click the X button (Remove Standards File).

4 (Optional) Repeat steps 2 and 3 if you want to remove additional standards files.

5 Click OK.

Command entry: STANDARDS

Toolbar: CAD Standards

To change the order of standards files associated with the current drawing

1 Click Manage tab ➤ Cad Standards panel ➤ Configure.

2 In the Configure Standards dialog box, Standards tab, select the standards file whose position you want to change in Standards Files Associated with the Current Drawing.

3 Do one of the following:

- Click the up arrow button (Move Up) to move the standards file up one position in the list.
- Click the down arrow button (Move Down) to move the standards file down one position in the list.

4 (Optional) Repeat steps 2 and 3 if you want to change the position of other standards files in the list.

5 Click OK.

Command entry: STANDARDS

Toolbar: CAD Standards

To specify which standards plug-ins to use when auditing a drawing

1 Click Manage tab ➤ Cad Standards panel ➤ Configure.

2 In the Configure Standards dialog box, Plug-ins tab, do one of the following:

- Select the check box for at least one plug-in to audit a drawing for standards violations.
- To select all plug-ins, right-click in the Plug-ins list. Click Select All. (You can clear all plug-ins. Right-click in the Plug-ins list. Click Clear All.)

3 Click OK.

Command entry: STANDARDS

Toolbar: CAD Standards

Quick Reference

Commands

STANDARDS

Manages the association of standards files with drawings.

System Variables

No entries.

Utilities

No entries

Command Modifiers

No entries

Check Drawings for Standards Violations

You can audit a drawing file for violations of standards and then fix violations. The Batch Standards Checker audits more than one file at a time.

After you associate a standards file with a drawing, you should periodically check the drawing to make sure it conforms with its standards. This is especially important when more than one person is updating the drawing file. For example, in a project with multiple subcontractors, one subcontractor may create new layers that do not comply with the standards you have defined. In that case, you need to be able to identify the nonstandard layers and fix them.

You can use the notification feature to alert users that a standards violation has occurred while working in a drawing. This feature allows you to fix a standards violation just after it has occurred, making it easier to create and maintain drawings that comply with standards.

Check a Single Drawing

You can use the CHECKSTANDARDS command to view all standards violations in the current drawing. The Check Standards dialog box reports each nonstandard object, along with any suggested fixes.

You can choose to fix or ignore each reported standards violation. If you ignore a reported violation, it is flagged in the drawing. You can turn off the display of ignored problems so that they are not reported as violations the next time you audit the drawing.

If no fix is provided for the current standards violation, no item is highlighted in the Replace With list, and the Fix button is not available. If you fix a standards violation that is currently displayed in the Check Standards dialog box, the violation is not removed from the dialog box until you click the Fix or the Next button.

Once the entire drawing has been audited, the Checking Complete message is displayed. This message summarizes the standards violations that were found in the drawing. The message also shows the violations that were fixed automatically, the violations that were fixed manually, and the violations that were ignored.

NOTE When a nonstandard layer contains multiple violations (for example, a violation where there is a nonstandard layer name and one for nonstandard layer properties), the first violation that is encountered is displayed. Subsequent violations that exist on the nonstandard layer are not evaluated, and therefore, not displayed. You will need to run the command again to check for additional violations.

Check Multiple Drawings

You can use the Batch Standards Checker to analyze multiple drawings and summarize standards violations in an HTML report. To run a batch standards audit, you must first create a standards check (CHX) file. The CHX file is a configuration and report file; it contains a list of drawing and standards files, as well as a report on a standards check.

By default, each drawing is checked against the standards files that are associated with it. Alternatively, you can override the default and choose another set of standards files to be used.

After the batch standards audit is complete, you can view an HTML report with details of the audit. You can also create notes that are included in the HTML report. This report can be exported and printed. In a collaborative environment, you can distribute the report to drafters so that they can fix any problems with their sections.

Use Notification of Standards Violations While Working in a Drawing

You can set options for notification in the CAD Standards Settings dialog box and with the STANDARDSVIOLATION system variable. If you select Display Alert upon Standards Violation in the dialog box, then an alert is displayed when a violation occurs while you work. If you select Display Standards Status

Bar Icon, an icon is displayed when you open a file associated with a standards file and when you create or modify nonstandard objects.

By default, if an associated standards file is missing or if a violation occurs while you work, a pop-up message appears in the lower-right corner of the application window (the status bar tray).

You should check a drawing for standards violations using the Check Standards dialog box before using the notification options. This prevents triggering a notification alert carried over from a previous session. After a drawing has been checked and fixed, the notification options trigger an alert only if a new violation occurs.

Display Alerts for Named Objects

If you select Display Alert upon Standards Violation, you are notified of violations when you create or edit only named objects (linetypes, text styles, layers, and dimensions). Standards violations that do not affect named objects do not trigger a notification alert. Also, if a named object was marked as ignored in the Check Standards dialog box, that object no longer triggers a notification alert, even though the named object is nonstandard. Changing nonstandard named objects, such as setting a nonstandard layer as current, does trigger a notification alert.

After an alert is displayed, you can choose to fix or not fix the violation. Choosing to fix the violation opens the Check Standards dialog box. If it is already open, the dialog box addresses the specific violations that just occurred. You can resume your previous work in the Check Standards dialog box after you respond to the most recent standards violations. Alternatively, if you do not want to fix violations when alerted, you can click Do Not Fix to dismiss the alert.

If you open a drawing that has one or more associated standards files, an Associated Standards File(s) icon is displayed in the status bar. If an associated standards file is missing, a Missing Standards File(s) icon is displayed in the status bar. If you double-click the Missing Standards File(s) icon and then resolve or dissociate missing standards files, the Missing Standards File(s) icon is replaced by the Associated Standards File(s) icon.

NOTE If you select Display Alert upon Standards Violation in the Check Standards dialog box and you fix a violation, you can return to a fixing operation where you left off. If you select Display Standards Status Bar icon and you click the icon and fix a violation, you must return to a standards-fixing operation from the beginning.

To audit a drawing for standards violations

1 Open a drawing that has one or more associated standards files.

An Associated Standards File(s) icon is displayed in the status bar. If an associated standards file is missing, a Missing Standards File(s) icon is displayed in the status bar.

NOTE If you click the Missing Standards File(s) icon and then resolve or dissociate missing standards files, the Missing Standards File(s) icon is replaced by the Associated Standards File(s) icon.

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2 In the drawing with one or more associated standards files, click Manage tab ➤ CAD Standards panel ➤ Check.

The Check Standards dialog box is displayed with the first standards violation reported under Problem.

3 Do one of the following:

- If you want to apply the item selected in the Replace With list to fix the violation reported under Problem, click Fix. If there is a recommended fix in the Replace With list, it is preceded by a check mark. If no recommended fix exists for the current standards violation, the Fix button is not available.
  The Check Standards dialog box automatically displays the next standards violation under Problem.
- Manually fix the standards violation. Click the Next button to display the next violation.
- Select Mark This Problem as Ignored. Click the Next button to display the next violation.
  Selecting Mark This Problem as Ignored flags the standards violation so you can prevent it from being displayed the next time you use the CHECKSTANDARDS command.
- Click the Next button to display the next violation.

4 Repeat steps 2 and 3 until you have cycled through all standards violations.

5 Click Close.

Command entry: CHECKSTANDARDS

Toolbar: CAD Standards

To turn the display of ignored problems on or off

1. In a drawing with one or more associated standards files, click Manage tab ➤ CAD Standards panel ➤ Check.
2. In the Check Standards dialog box, click Settings.
3. In the CAD Standards Settings dialog box, select or clear Show Ignored Problems. Click OK.

Command entry: CHECKSTANDARDS

Toolbar: CAD Standards

To turn notification for standards violations on or off

1. Click Manage tab ➤ Cad Standards panel ➤ Configure.
2. In the Configure Standards dialog box, click Settings.
3. Do one of the following:
   - To turn off standards notifications, select Disable Standards Notifications.
   - To be notified by alert, select Display Alert upon Standards Violation.
   - To be notified by the display of an icon in the status bar, select Display Standards Status Bar Icon.
4. Click OK.

Command entry: STANDARDS

Toolbar: CAD Standards

To start the Batch Standards Checker

- Click Start menu (Windows) ➤ All Programs ➤ Autodesk ➤ ProductNameLong ➤ Batch Standards Checker.

NOTE When you start the Batch Standards Checker from a DOS command line using the DWGCHECKSTANDARDS command, the Batch Standards Checker accepts a variety of command-line parameters that can be used for scripting batch standards audits.

To create a standards check file for the Batch Standards Checker

1. Start the Batch Standards Checker.
2. In the Batch Standards Checker, click File menu ➤ New Check File. (You can also click New on the Batch Standards Checker toolbar.)
3. On the Drawings tab, click the + button (Add Drawing).
4. In the Open dialog box, select a drawing to audit.
5. (Optional) Repeat steps 3 and 4 if you want to add additional drawings to the standards check file.
6. Click File menu ➤ Save As. (You can also click the Save As button on the Batch Standards Checker toolbar.)
7. In the Save As dialog box, in File Name, enter a name for the file.
8. Click Save.

To open an existing standards check file

1. Start the Batch Standards Checker.
2. In the Batch Standards Checker, click File menu ➤ Open Check File. (You can also click Open on the Batch Standards Checker toolbar.)
3. In the Open dialog box, locate and select a standards check file.
4. Click Open.

To specify a standards override for a standards check file

1. Start the Batch Standards Checker.
2. Create a standards check file or open an existing one.

3 In the Batch Standards Checker, Standards tab, select the Check All Drawings Using the Following Standards Files option.

4 Click the + button (Add Standards File).

5 In the Open dialog box, select a standards file to use as an override. Click Open.

6 (Optional) Repeat steps 5 and 6 if you want to add additional standards files as overrides.

7 In the Batch Standards Checker, click File menu ➤ Save Check File. (You can also click Save on the Batch Standards Checker toolbar.)

To audit a set of drawings for standards violations

1 Start the Batch Standards Checker.

2 Open an existing standards check file, or create a new one.

3 In the Batch Standards Checker, click Check menu ➤ Start Check. (You can also click Start Check on the Batch Standards Checker toolbar.)

When the batch audit is complete, the report is displayed in a browser window.

You can cancel the batch audit at any time by click Stop Check.

NOTE You cannot check encrypted files using the Batch Standards Checker.

To add notes to the batch audit report

1 Start the Batch Standards Checker.

2 Create a standards check file or open an existing one.

3 In the Batch Standards Checker, Notes tab, enter notes to be included in the report.

4 Click File menu ➤ Save Check File. (You can also click Save on the Batch Standards Checker toolbar.)

To view a previously generated batch audit report

1 Start the Batch Standards Checker.

2 Open the standards check file that generated the report you want to view.

3 In the Batch Standards Checker, click Check menu ➤ View Report.(You can also click View Report on the Batch Standards Checker toolbar.)

   The report is displayed in a browser window.

4 Select one of the following options to filter the data displayed in the report:

   - **Overview.** Summarizes the number of problems encountered in each audited drawing.
   - **Plug-ins.** Summarizes the standards plug-ins installed on the system used to run the batch audit.
   - **Standards.** Summarizes the standards files used to check the drawings in the batch audit.
   - **Problems.** Provides detailed information about each problem that was encountered.
   - **Ignored Problems.** Provides detailed information about problems that were flagged as ignored.
   - **All.** Shows all available standards audit information.

5 In the For list, select individual drawings to view their audit information.

To export a batch audit report

1 Start the Batch Standards Checker.

2 Create a standards check file or open an existing one.

3 In the Batch Standards Checker, click Check ➤ Export Report. (You can also click Export Report on the Batch Standards Checker toolbar.)

Quick Reference

Commands

CHECKSTANDARDS

Checks the current drawing for standards violations.

STANDARDS

Manages the association of standards files with drawings.

System Variables

STANDARDSVIOLATION

Specifies whether a user is notified of standards violations in the current drawing when a nonstandard object is created or modified.

Utilities

Batch Standards Checker

Audits a set of drawings for standards violations.

Command Modifiers

No entries

Translate Layer Names and Properties

You can reorganize a drawing's layers to match a set of layer standards.

Convert Layers to Established Drawing Standards

With the Layer Translator, you can convert layers in one drawing to layer standards that you define.

For example, if you receive a drawing from a company that does not follow your company's layer standards, you can convert the drawing's layer names and properties to your company's standards. You can map layers in the drawing you are currently working on to different layers in another drawing or standards file, and then convert the current layers using those mappings. If the drawings contain layers of the same name, the Layer Translator can automatically change the properties of the current layers to match those in the other layers.

You can save your layer translation mappings in a file and reuse them with other drawings.

View Selected Drawing Layers

With the Layer Translator, you can control which layers are visible in the drawing area.

You can choose to display objects on all layers in your drawing, or objects on layers you specify. By viewing selected layers, you can visually verify the contents of those layers.

Purge Unreferenced Layers

With the Layer Translator, you can purge (delete) all unreferenced layers from a drawing.

For example, if your drawing includes layers that you do not need, you may want to remove those layers. Reducing the number of layers makes working with the remaining layers more manageable.

To convert a drawing's layers to standard layer settings

1 Click Manage tab ➤ CAD Standards panel ➤ Layer Translator. 

2 In the Layer Translator, do one of the following:

- Click Load to load layers from a drawing, a drawing template, or a drawing standards file. In the Select Drawing File dialog box, select the file you want. Click Open.
- Click New to define a new layer. In the New Layer dialog box, enter a name for the new layer and select its properties. Click OK.

You can repeat step 2 as many times as you want. If you load other files containing layers of the same name as ones already shown in the Translate To list, the layer properties first loaded in the list are maintained; any duplicate layer properties are ignored.

3 Map layers in the current drawing to the layers you want to convert to. Use either or both of the following methods to map layers:

- To map all identically named layers from one list to the other, click Map Same.
- To map individual layers, in the Translate From list, select one or more layers. In the Translate To list, select the layer whose properties you want to use. Click Map to define the mapping. You can repeat this method for each layer or group of layers to be translated.

To remove a mapping, select the mapping from the Layer Translation Mappings list. Click Remove. To remove all mappings, right-click in the list. Click Remove All.

4 (Optional) You can perform the following tasks in the Layer Translator:

- To change the properties of a mapped layer, in the Layer Translation Mappings list, select the mapping whose properties you want to change. Click Edit. In the Edit Layer dialog box, change the linetype, color, lineweight, or plot style of the mapped layer. Click OK.

- To customize the process of layer translation, click Settings. In the Settings dialog box, select the options you want. Click OK.
- To save layer mappings to a file, click Save. In the Save Layer Mappings dialog box, enter a file name. Click OK.

5 Click Translate to perform the layer translations you have specified.

Command entry: LAYTRANS

Toolbar: CAD Standards

To specify which layers are displayed in the drawing area

1 Click Manage tab ➤ CAD Standards panel ➤ Layer Translator.

2 In the Layer Translator, click Settings.

3 In the Settings dialog box, do one of the following:
   - To display objects on selected layers in your drawing, select Show Layer Contents When Selected. Only the layers selected in the Layer Translator dialog box are displayed in the drawing area.
   - To display objects on all layers in your drawing, clear Show Layer Contents When Selected.

4 Click OK.

Command entry: LAYTRANS

Toolbar: CAD Standards

To purge all unreferenced layers from a drawing

1 Click Manage tab ➤ CAD Standards panel ➤ Layer Translator.

2 In the Layer Translator, right-click in the Translate From list. Click Purge Layers. All unreferenced layers are deleted from the current drawing.

Command entry: LAYTRANS

Toolbar: CAD Standards 

Quick Reference

Commands

LAYTRANS

Translates the layers in the current drawing to specified layer standards.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Control the Drawing Views

Change Views

9

You can magnify the details in your drawing for a closer view or shift the view to a different part of the drawing. If you save views by name, you can restore them later.

Pan or Zoom a View

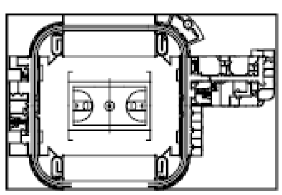
You can pan to reposition the view in the drawing area or zoom to change magnification.

With the Realtime option of PAN, you pan dynamically by moving your pointing device. Like panning with a camera, PAN does not change the location or magnification of objects on your drawing; it changes only the view.

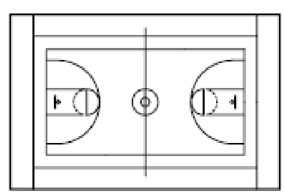
You can change the magnification of a view by zooming in and out, which is similar to zooming in and out with a camera. ZOOM does not change the absolute size of objects in the drawing; it changes only the magnification of the view.

When you work with minute parts in your drawing, you may need to zoom out frequently to see an overview of your work. Use ZOOM Previous to return quickly to the prior view.

The options described here are the options most commonly used. For a description of all ZOOM options, see the *Command Reference*.



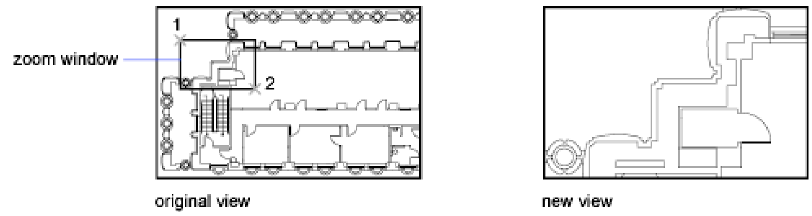
zoomed out



zoomed in

Zoom to Magnify a Specified Rectangular Area

You can quickly zoom on a rectangular area of your drawing by specifying two diagonal corners of the area you are interested in.



The lower-left corner of the area you specify becomes the lower-left corner of the new display. The shape of the zoom area you specify does not correspond exactly to the new view, which must fit the shape of the viewport.

Zoom in Real Time

With the Realtime option, you zoom dynamically by moving your pointing device up or down. By right-clicking, you can display a shortcut menu with additional viewing options.

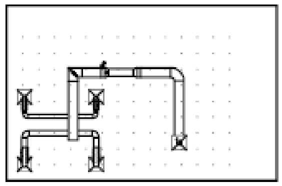
Zoom to Magnify One or More Objects

ZOOM Objects displays a view with the largest possible magnification that includes all of the objects you selected.

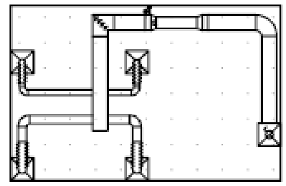
Zoom to View All Objects in the Drawing

ZOOM Extents displays a view with the largest possible magnification that includes all of the objects in the drawing. This view includes objects on layers that are turned off but does not include objects on frozen layers.

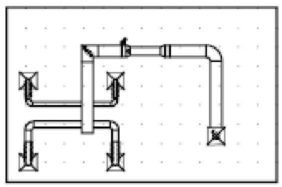
ZOOM All displays either the user-defined grid limits or the drawing extents, whichever view is larger.



current view



zoomed to extents



zoomed to show entire drawing (all)

See also:

- Pointing Devices
- Scale Views in Layout Viewports on page 456
- Toolbars on page 32

To pan by dragging

1 Click View tab ➤ Navigate panel ➤ Pan. 

2 When the hand cursor is displayed, click and hold your pointing device as you move.

NOTE If you are using a wheel mouse, hold down the wheel button and move the mouse.

---

3 To exit, press Enter or Esc, or right-click.

Command entry: PAN

Toolbar: Standard 

To pan by specifying points

1 Click View menu ➤ Pan ➤ Point.

2 Specify a base point. This is the point you want to change.

3 Specify a second (pan to) point. This is the new location for the point you selected first.

To zoom by dragging

1 Click View tab ➤ Navigate panel ➤ Realtime.

2 When the magnifying glass cursor is displayed, click and hold your pointing device and drag vertically to zoom in and out.

3 To exit, press Enter or Esc, or right-click.

Command entry: ZOOM

Toolbar: Standard

To zoom in to an area by specifying its boundaries

1 Click View tab ➤ Navigate panel ➤ Window.

2 Specify one corner of the rectangular area you want to view.

3 Specify the opposite corner.

   You automatically choose the Window option when you specify a point immediately after starting the ZOOM command.

Command entry: ZOOM

Toolbar: Standard

Shortcut menu: With the ZOOM command active, right-click in the drawing area. Click Zoom Window.

To display the drawing extents by zooming

- Click View tab ➤ Navigate panel ➤ Extents. 

All objects in the drawing are displayed to be as large as possible and still fit in the current viewport or the drawing area.

Toolbar: Standard 
Command entry: ZOOM
Shortcut menu: With the ZOOM command active, right-click in the drawing area. Click Zoom Extents.

To display the area of the grid limits by zooming

- Click View tab ➤ Navigate panel ➤ All. 

The limits of the drawing grid fill the current viewport or the drawing area. If there are any objects outside the grid limits, they are also included.

Command entry: ZOOM

Toolbar: Standard 

To restore the previous view

- Click View tab ➤ Navigate panel ➤ Previous. 

Zoom Previous restores only the view magnification and position, not the previous content of an edited drawing.

Command entry: ZOOM

Toolbar: Standard 
Shortcut menu: With the ZOOM command active, right-click in the drawing area. Click Zoom Original.

Quick Reference

Commands

DSVIEWER

Opens the Aerial View window.

PAN

Moves the view in the current viewport.

UNDO

Reverses the effect of commands.

VIEWRES

Sets the resolution for objects in the current viewport.

VTOPTIONS

Displays a change in view as a smooth transition.

ZOOM

Increases or decreases the magnification of the view in the current viewport.

System Variables

EXTMAX

Stores the upper-right point of the drawing extents.

EXTMIN

Stores the lower-left point of the drawing extents.

MBUTTONPAN

Controls the behavior of the third button or wheel on the pointing device.

RTDISPLAY

Controls the display of raster images and OLE objects during Realtime ZOOM or PAN.

VTENABLE

Controls when smooth view transitions are used.

VTDURATION

Sets the duration of a smooth view transition, in milliseconds.

VTFPS

Sets the minimum speed of a smooth view transition, in frames per second.

WHIPARC

Controls whether the display of circles and arcs is smooth.

WHIPTHREAD

Controls whether to use an additional processor to improve the speed of operations such as ZOOM that redraw or regenerate the drawing.

ZOOMFACTOR

Controls how much the magnification changes when the mouse wheel moves forward or backward.

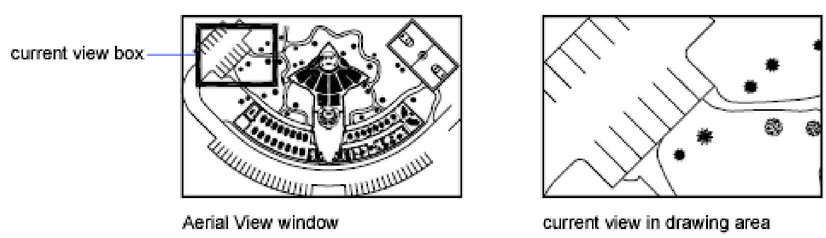
Pan and Zoom with the Aerial View Window

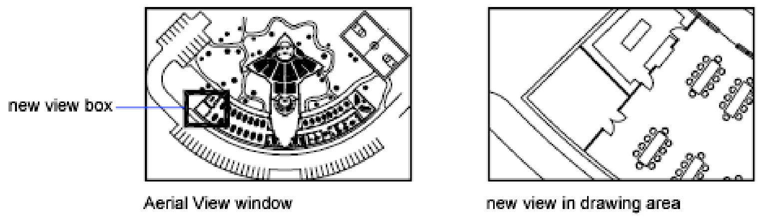
In a large drawing, you can pan and zoom quickly in a window that can display the whole drawing.

You can use the Aerial View window to change the view in your current viewport quickly. If you keep the Aerial View window open as you work, you can zoom and pan without interrupting your current command. You can also specify a new view without having to choose a menu option or enter a command.

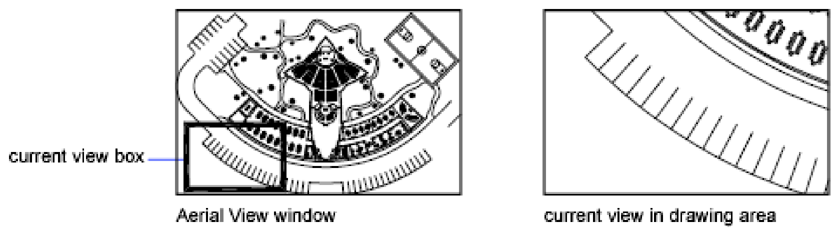
Use the View Box to Pan and Zoom

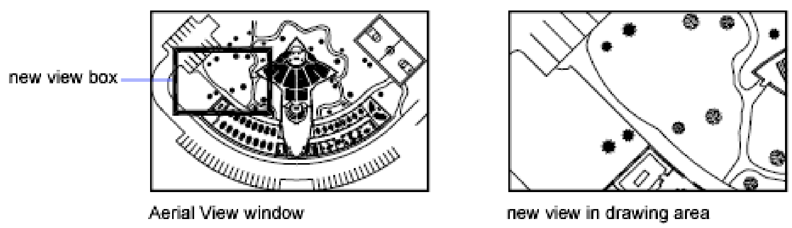
Within the Aerial View window is a view box, a heavy rectangle that displays the boundary of the view in your current viewport. You can change the view in the drawing by changing the view box in the Aerial View window. To zoom in to the drawing, make the view box smaller. To zoom out of the drawing, make the view box larger. All pan and zoom operations are performed by left-clicking. Right-click to end a pan or zoom operation.





To pan the drawing, move the view box.





Change the View Displayed Inside the Aerial View Window

You can use the Aerial View toolbar buttons to change the magnification of the image in the Aerial View window, or you can resize the image incrementally. These changes do not affect the view in the drawing itself.

Use the Aerial View Window with Multiple Viewports

Only the view in the current viewport is used in the Aerial View window.

The Aerial View image is updated as you make changes to your drawing and as you select different viewports. In complex drawings, you may want to turn off this dynamic updating to improve drawing speed. If you turn off this feature, the Aerial View image is updated only when you activate the Aerial View window.

To zoom to a new area using the Aerial View window

1 Click View menu ➤ Aerial View.

2 In the Aerial View window, click inside the view box until you see the arrow.

3 Drag to the right to zoom out. Drag to the left to zoom in.

4 Right-click to end the zoom operation.

Command entry: DSVIEWER

To pan using the Aerial View window

1 Click View menu ➤ Aerial View.

2 In the Aerial View window, click inside the view box until you see an X.

3 Drag to change the view.

4 Right-click to end the pan operation.

Command entry: DSVIEWER

To display the entire drawing in the Aerial View window

1 Click View menu ➤ Aerial View.

2 In the Aerial View window, click View menu ➤ Global.

Command entry: DSVIEWER

To increase or decrease magnification of the Aerial View image

1 Click View menu ➤ Aerial View.

2 On the Aerial View toolbar, click Zoom Out or Zoom In.

NOTE 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.

Command entry: DSVIEWER

To turn dynamic updating on and off

- Click View menu ➤ Aerial View.
- In the Aerial View window, click Options menu ➤ Dynamic Update.
  A check mark indicates that the Aerial View window shows changes as they occur.

To turn viewport updating on and off

- Click View menu ➤ Aerial View.
- In the Aerial View window, click Options menu ➤ Auto Viewport.
  A check mark indicates that Aerial View displays the current viewport as you switch viewports.

Quick Reference

Commands

DSVIEWER

Opens the Aerial View window.

Save and Restore Views

When you save specific views by name, you can restore them for layout and plotting or when you need to refer to specific details. You can create and save views using the VIEW or the CAMERA command.

A named view created with the VIEW command consists of a specific magnification, position, and orientation. In each drawing session, you can restore the last view displayed in each viewport and up to 10 previous views.

Named views and cameras are saved with a drawing and can be used any time. When you are composing a layout, you can restore a named view or a camera to a viewport on the layout.

Save a View

When you name and save a view, the following settings are saved:

- Magnification, center point, and view direction
- View category that you assign to the view (optional)

- The location of the view (the Model tab or a specific layout tab)
- Layer visibility in the drawing at the time the view is saved
- User coordinate system
- 3D perspective
- Live section
- Visual style
- Background

Restore a Named View

You restore a named view to the current viewport. You can use named views to do the following:

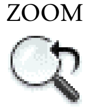
- Compose a layout.
- Restore a view that you use frequently while you are working in model space.
- Control which model space view is displayed when the drawing is opened.

To restore the previous view

- Click View tab ➤ Navigate panel ➤ Previous. 

Zoom Previous restores only the view magnification and position, not the previous content of an edited drawing.

Command entry: ZOOM

Toolbar: Standard 
Shortcut menu: With the ZOOM command active, right-click in the drawing area. Click Zoom Original.

To display the previous view during PAN Realtime and ZOOM Realtime

- Click View tab ➤ Navigate panel ➤ Zoom drop-down ➤ Previous.
  

  Alternatively, right-click in the drawing area. Click Zoom Original.

To save and name a view

1 Do one of the following:
   - If you have more than one viewport in model space, click inside the viewport that contains the view you want to save.
   - If you are working in a layout, select the viewport.

2 Click View tab ➤ Views panel ➤ Named Views. 

3 In the View Manager, click New.

4 In the New View dialog box, View Name box, enter a name for the view.

   If the drawing is part of a sheet set, the view categories for the sheet set are listed. You can add a category or select one from the list.

5 In the Boundary section, select one of the following options to define the area of the view:
   - **Current Display.** Includes all of the drawing that is currently visible.
   - **Define Window.** Saves part of the current display. The dialog box closes while you use the pointing device in the drawing to specify opposite corners of the view. To redefine the window, click the Define View Window button.

6 Click OK twice to save the new view.

Command entry: VIEW

Toolbar: View 

To restore a saved view

1 Do one of the following:
   - If you have more than one viewport in model space, click inside the viewport that contains the view you want to restore.
   - If you are working in a layout, select the viewport.

2 Click View tab ➤ Views panel ➤ Named Views. 

3 In the View Manager, Views list, select the view that you want to restore.

4 Click Set Current. Click OK.

Command entry: VIEW

Toolbar: View 

To rename a view

1 Click View tab ➤ Views panel ➤ Named Views. 

2 In the View Manager, click the view name you want to change. If the view isn't already displayed, expand the appropriate View list, and then click a view name.

3 In the General section of the Properties panel, select the view name. Enter a new name.

4 Click OK.

Command entry: VIEW

Toolbar: View 

To change the properties of a view

1 Click View tab ➤ Views panel ➤ Named Views. 

2 In the View Manager, click the view name you want to change. If the view isn't already displayed, expand the appropriate View list, and then click a view name.

3 In the Properties panel, click the property that you want to change.

4 Specify the new property value by entering a new value or by selecting from a list of values. Click OK.

Command entry: VIEW

Toolbar: View

To delete a named view

1 Click View tab ➤ Views panel ➤ Named Views.

2 In the View Manager, click a view, and then click Delete.

3 Click OK.

Command entry: VIEW

Toolbar: View

To view a list of saved viewport arrangements

- Click View tab ➤ Viewports panel ➤ Named.

The list of saved viewports is displayed in the Viewports dialog box, Named Viewports tab.

Command entry: VPORTS

Toolbar: Viewports

Quick Reference

Commands

VIEW

Saves and restores named views, camera views, layout views, and preset views.

VPORTS

Creates multiple viewports in model space or paper space.

ZOOM

Increases or decreases the magnification of the view in the current viewport.

Control the 3D Projection Style

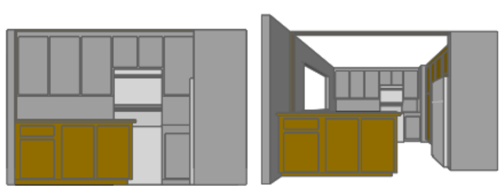
You can view both parallel and perspective projection of a 3D model.

Overview of Parallel and Perspective Views

You can create realistic visual effects in a drawing by defining either parallel or perspective projections of a model.

The difference between perspective views and parallel projections is that perspective views require a distance between a theoretical camera and target point. Small distances produce severe perspective effects; large distances produce mild effects.

The following illustration shows the same model in both a parallel projection and perspective projection. Both are based on the same viewing direction.



Parallel projection   Perspective projection

Quick Reference

Commands

3DORBIT

Rotates the view in 3D space, but constrained to horizontal and vertical orbit only.

DVIEW

Defines parallel projection or perspective views by using a camera and target.

System Variables

BACKZ

Stores the back clipping plane offset from the target plane for the current viewport, in drawing units.

FRONTZ

Stores the front clipping plane offset from the target plane for the current viewport, in drawing units.

LENSLENGTH

Stores the length of the lens (in millimeters) used in perspective viewing.

TARGET

Stores the location (as a UCS coordinate) of the target point for the current viewport.

VIEWDIR

Stores the viewing direction in the current viewport, expressed in UCS coordinates.

VIEWMODE

Stores the View mode for the current viewport.

VIEWTWIST

Stores the view rotation angle for the current viewport measured relative to the WCS.

WORLDVIEW

Define a Perspective Projection (DVIEW)

Perspective projections require a distance between a theoretical camera and a target point. Small distances produce severe perspective effects; large distances produce milder effects.

A perspective view remains in effect until the perspective effect is turned off or until a new view is defined in its place.

To define a perspective view of a 3D model using DVIEW

1. At the Command prompt, enter **dview**.
2. Select the objects to display.
3. Enter **ca** (Camera).

   By default, a camera point is set at the center of the drawing.
4. Adjust the view as if you're aiming a camera.

   A representation of a house shows the current viewing angle. You can set your view dynamically by moving the crosshairs and clicking.
5. To switch between angle-input methods, enter **t** (Toggle Angle).

   You also can adjust the view with one of two angle-input methods.

   - For the Enter Angle from the XY Plane option, enter the angle of the camera up or down relative to the *XY* plane of the current UCS. The default setting, 90 degrees, points the camera straight down from above.
     After you enter the angle, the camera is locked at that height, and you can rotate the camera about the target with the rotation angle measured relative to the *X* axis of the current UCS.
   - For the Enter Angle in the XY Plane from the X Axis option, rotate the camera about the target with the rotation angle measured relative to the *X* axis of the current UCS.
6. To turn on the perspective view, enter **d** (Distance).
7. Specify a distance, or press Enter to set the perspective view.

   You can use the slider bar to set the distance between the selected objects and the camera, or you can enter a real number. If the target and camera points are very close (or if the Zoom option is set high), you might see only a small part of your drawing.

Command entry: DVIEW

To turn off a perspective view using DVIEW

1. At the Command prompt, enter **dview**.
2. Select the objects to display.
3. Enter **o** (Off).

   Perspective is turned off and the view is restored to a parallel projection.

Command entry: DVIEW

Quick Reference

Commands

3DORBIT

Rotates the view in 3D space, but constrained to horizontal and vertical orbit only.

DVIEW

Defines parallel projection or perspective views by using a camera and target.

System Variables

BACKZ

Stores the back clipping plane offset from the target plane for the current viewport, in drawing units.

FRONTZ

Stores the front clipping plane offset from the target plane for the current viewport, in drawing units.

LENSLENGTH

Stores the length of the lens (in millimeters) used in perspective viewing.

PERSPECTIVE

Specifies whether the current viewport displays a perspective view.

PERSPECTIVECLIP

Determines the location of eyepoint clipping.

TARGET

Stores the location (as a UCS coordinate) of the target point for the current viewport.

VIEWDIR

Stores the viewing direction in the current viewport, expressed in UCS coordinates.

VIEWMODE

Stores the View mode for the current viewport.

VIEWTWIST

Stores the view rotation angle for the current viewport measured relative to the WCS.

WORLDVIEW

Define a Parallel Projection

You can define a parallel projection.

To determine the point or angle in model space, you can

- Choose a preset 3D view from the View toolbar.
- Enter a coordinate or angles that represent your viewing location in 3D.
- Change to a view of the *XY* plane of the current UCS, a saved UCS, or the WCS.
- Change the 3D view dynamically with your pointing device.
- Set front and back clipping planes to limit the objects being displayed.

Viewing in 3D is available only in model space. If you are working in paper space, you cannot use 3D viewing commands such as VPOINT, DVIEW, or PLAN to define paper space views. The view in paper space is always a plan view.

To create a parallel projection dynamically

1 At the Command prompt, enter **dview**.

2 Do *one* of the following:
   - Select the objects to display and press Enter.
   - Press Enter without selecting objects to see a representation of a house that shows the current viewing angle.
3 Enter **ca** (Camera).
   By default, a camera point is set at the center of the drawing.
4 Adjust the view as if you're aiming a camera.
   You can set your view dynamically by moving the crosshairs and clicking.
5 To switch between the angle-input methods, enter **t** (Toggle Angle).
   You can also adjust the view with one of two angle-input methods.
   - For the Enter Angle from the XY Plane option, enter the angle of the camera up or down relative to the *XY* plane of the current UCS. The default setting, 90 degrees, points the camera straight down from above.
     After you enter the angle, the camera is locked at that height, and you can rotate the camera about the target with the rotation angle measured relative to the *X* axis of the current UCS.
   - For the Enter Angle in the XY Plane from the X Axis option, rotate the camera about the target with the rotation angle measured relative to the *X* axis of the current UCS.
6 When you finish specifying the parallel projection, press Enter.

Command entry: DVIEW

Quick Reference

Commands

DVIEW

Defines parallel projection or perspective views by using a camera and target.

PLAN

Displays the plan view of a specified user coordinate system

VPOINT

Sets the viewing direction for a 3D visualization of the drawing.

System Variables

PERSPECTIVE

Specifies whether the current viewport displays a perspective view.

VIEWDIR

Stores the viewing direction in the current viewport, expressed in UCS coordinates.

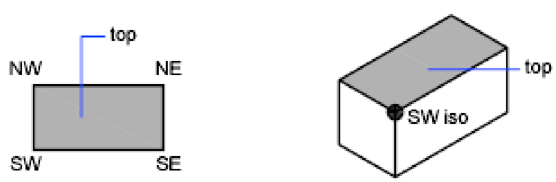
WORLDVIEW

Choose Preset 3D Views

You can select predefined standard orthographic and isometric views by name or description.

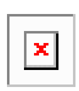
A quick way to set a view is to choose one of the predefined 3D views. You can select predefined standard orthographic and isometric views by name or description. These views represent commonly used options: Top, Bottom, Front, Left, Right, and Back. In addition, you can set views from isometric options: SW (southwest) Isometric, SE (southeast) Isometric, NE (northeast) Isometric, and NW (northwest) Isometric.

To understand how the isometric views work, imagine you are looking down at the top of a box. If you move toward the lower-left corner of the box, you are viewing the box from the SW Isometric View. If you move toward the upper-right corner of the box, you are viewing it from NE Isometric View.



To use a preset 3D view

- Click View tab ➤ Views panel ➤ Named Views.
- Select a preset view (Top, Bottom, Left, and so on).

Toolbar: View 
Command entry: VIEW

Quick Reference

Commands

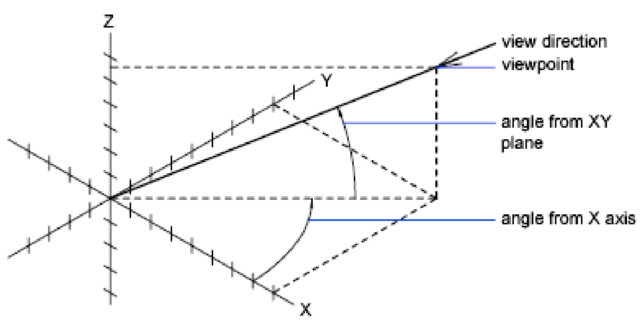
VIEW

Define a 3D View with Coordinate Values or Angles

You can define a viewing direction by entering the coordinate values of a point or the measures of two angles of rotation.

This point represents your position in 3D space as you view the model while looking toward the origin (0,0,0). Viewpoint coordinate values are relative to the world coordinate system unless you change the WORLDVIEW system variable. The conventions for defining standard views differ between architectural (AEC) and mechanical design. In AEC design, the perpendicular view of the *XY* plane is the top or plan view; in mechanical design, the perpendicular view of the *XY* plane is the front view.

You can rotate a view using DDVPOINT. The following illustration shows a view defined by two angles relative to the *X* axis and the *XY* plane of the WCS.



To set a view with a viewpoint coordinate

1 Click View menu ➤ 3D Views ➤ Viewpoint.

2 Click inside the compass to specify the viewpoint. The selected viewpoint is used to view the drawing in the direction of 0,0,0.

Command entry: VPOINT

To set a view with two angles of rotation

1. Click View menu ➤ 3D Views ➤ Viewpoint.
2. Enter r (Rotate) to specify a new direction using two angles.
3. Enter an angle in the *XY* plane measured from the positive *X* axis.
4. Enter an angle from the *XY* plane that represents your position while viewing the model in the direction of 0,0,0.

Command entry: VPOINT

To set standard views with VPOINT (AEC convention)

1. Click View menu ➤ 3D Views ➤ Viewpoint.
2. Enter a coordinate according to the viewpoint you want:
   - Enter **0,0,1** for a top (plan) view.
   - Enter **0,-1,0** for a front view.
   - Enter **1,0,0** for a right side view.
   - Enter **1,-1,1** for an isometric view.

Command entry: VPOINT

To set standard views with VPOINT (mechanical design convention)

1. Click View menu ➤ 3D Views ➤ Viewpoint.
2. Enter a coordinate according to the viewpoint you want:
   - Enter **0,1,0** for a top view.
   - Enter **0,0,1** for a front view.
   - Enter **1,0,0** for a right side view.
   - Enter **1,1,1** for an isometric view. This view is identical to one with a right-out of 45 degrees and a top-out of 35.267 degrees.

Command entry: VPOINT

Quick Reference

Commands

VIEW

VPOINT

Sets the viewing direction for a 3D visualization of the drawing.

System Variables

WORLDVIEW

Change to a View of the XY Plane

You can change the current viewpoint to a plan view of the current UCS, a previously saved UCS, or the WCS.

A plan view is a view aimed toward the origin (0,0,0) from a point on the positive *Z* axis. This results in a view of the *XY* plane.

You can restore the view and coordinate system that is the default for most drawings by setting the UCS orientation to World and then setting the 3D view to Plan View.

To change the current view to the *XY* plane

1. Click View menu ➤ 3D Views ➤ Plan View.
2. Select one of the following options:
   - Current (for the current UCS)
   - World (for the WCS)
   - Named (for a saved UCS)

NOTE PLAN changes the viewing direction and turns off perspective and clipping; it does not change the current UCS. Any coordinates entered or displayed subsequent to the PLAN command remain relative to the current UCS.

Command entry: PLAN

Quick Reference

Commands

DVIEW

Defines parallel projection or perspective views by using a camera and target.

PLAN

Displays the plan view of a specified user coordinate system

System Variables

BACKZ

Stores the back clipping plane offset from the target plane for the current viewport, in drawing units.

FRONTZ

Stores the front clipping plane offset from the target plane for the current viewport, in drawing units.

VIEWDIR

Stores the viewing direction in the current viewport, expressed in UCS coordinates.

VIEWTWIST

Stores the view rotation angle for the current viewport measured relative to the WCS.

WORLDVIEW

Shade a Model and Use Edge Effects

Hiding lines enhances the drawing and clarifies the design. The addition of shading produces a more realistic image of your model.

Use a Visual Style to Display Your Model

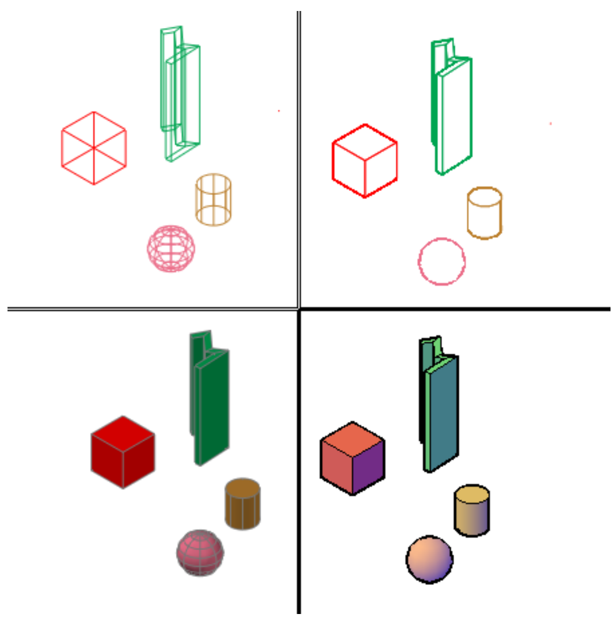
A visual style is a collection of settings that control the display of edges and shading in the viewport. Instead of using commands and setting system variables, you change the properties of the visual style. As soon as you apply a visual style or change its settings, you can see the effect in the viewport.

The Visual Styles Manager displays sample images of the visual styles available in the drawing. The selected visual style is indicated by a yellow border, and its settings are displayed in the panel below the sample images.

When the ribbon on page 22 is displayed, you can change some frequently used settings directly or open the Visual Styles Manager.

Five default visual styles are supplied with the product:

- 2D Wireframe. Displays the objects using lines and curves to represent the boundaries. Raster and OLE objects, linetypes, and lineweights are visible.
- 3D Wireframe (upper left in the illustration). Displays the objects using lines and curves to represent the boundaries.
- 3D Hidden (upper right). Displays the objects using 3D wireframe representation and hides lines representing back faces.
- Realistic (lower left). Shades the objects and smooths the edges between polygon faces. Materials that you have attached to the objects are displayed.
- Conceptual (lower right). Shades the objects and smooths the edges between polygon faces. Shading uses the Gooch face style, a transition between cool and warm colors rather than dark to light. The effect is less realistic, but it can make the details of the model easier to see.



In shaded visual styles, faces are lighted by two distant light sources that follow the viewpoint as you move around the model. This default lighting is designed to illuminate all faces in the model so that they are visually discernable. Default lighting is available only when other lights, including the sun, are off.

You can select a visual style and change its settings at any time. The changes you make are reflected in the viewports where the visual style is applied. For more information about face settings, environment settings, and edge settings, see Customize a Visual Style on page 298. Any changes you make to a visual style are saved in the drawing.

To save a visual style in the drawing

- At the Command prompt, enter **vssave**.

To apply a visual style to a viewport

1. Click View tab ➤ 3D Palettes panel ➤ Visual Styles Manager. 
2. Click in the viewport to make it current.
3. In the Visual Styles Manager, double-click the sample image of the visual style.

   The selected visual style is applied to the model in the viewport.

   An icon in the sample image indicates that the visual style is in use in the current viewport. When you change viewports, the icon changes to indicate that the visual style is in use in the current drawing.

   Visual Styles 

   Right-click the visual style in the Visual Styles Manager. Click Apply to Current Viewport.

   VISUALSTYLES

To store a visual style on a tool palette

1. Click View tab ➤ 3D Palettes panel ➤ Visual Styles Manager. 

2 Click View tab ➤ Palettes panel ➤ Tool Palettes.

3 In the Tool Palettes window, click the Visual Styles tab.

4 In the Visual Styles Manager, select the sample image of the visual style.

5 Below the images, click the Export the Selected Visual Style to the Tool Palette button.

Command entry: VISUALSTYLES, TOOLPALETTES

Toolbar: Visual Styles
Shortcut menu: Right-click the visual style in the Visual Styles Manager. Click Export to Active Tool Palette.

To use a visual style from a different drawing

1 Open the drawing that has the visual style that you want to use.

2 Click View tab ➤ 3D Palettes panel ➤ Visual Styles Manager.

3 Click View tab ➤ Palettes panel ➤ Tool Palettes.

4 In the Tool Palettes window, click the Visual Styles tab.

5 In the Visual Styles Manager, select the sample image of the visual style.

6 Below the images, click the Export the Selected Visual Style to the Tool Palette button.

7 Open the drawing where you want to use the visual style.

8 On the tool palette, select the visual style.

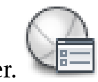
9 Right-click and click Add to Current Drawing.

The visual style is added to the sample images in the Visual Styles Manager and the ribbon.

Command entry: VISUALSTYLES, TOOLPALETTES

Toolbar: Visual Styles 
Shortcut menu: Right-click the visual style in the Visual Styles Manager. Click Export to Active Tool Palette.

To restore the original settings of a default visual style

1. Click View tab ➤ 3D Palettes panel ➤ Visual Styles Manager. 
2. Select a default visual style.

   A default visual style displays a drawing icon at bottom right in the sample image.
3. Right-click and click Reset to Default.

Command entry: VISUALSTYLES

Toolbar: Visual Styles 

Quick Reference

Commands

SHADEMODE

Starts the VSCURRENT command.

TOOLPALETTES

Opens the Tool Palettes window.

VISUALSTYLES

Creates and modifies visual styles and applies a visual style to a viewport.

VISUALSTYLESCLOSE

Closes the Visual Styles Manager.

System Variables

VSLIGHTINGQUALITY

Sets the lighting quality in the current viewport.

Customize a Visual Style

You can create your own visual styles by changing the face and edge settings and using shadows and backgrounds.

Shade and Color Faces

Shading and color effects control the display of faces in a model.

Face Styles

The face style defines the shading on a face. Real (below left) is meant to produce the effect of realism. Gooch (below right) can show details better by softening the contrast between lighted areas and shadowed areas. Lighted areas use warm tones and darker areas cool tones.

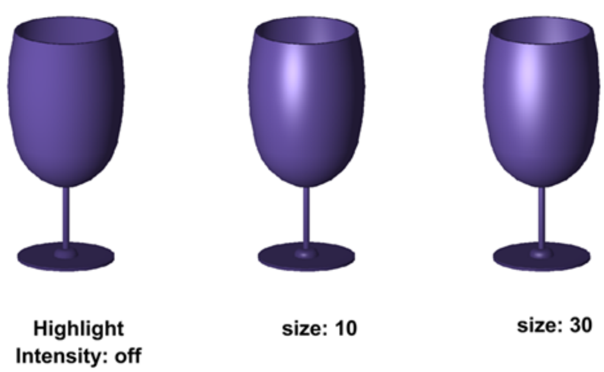
When the None face style is active, there is no shading, and only edges are displayed if Edge Mode is set to Facet Edges or Isolines under Edge Settings.

Lighting Quality

Faceted lighting computes a single color for each face. Objects appear flatter. Smooth lighting smoothes the edges between polygon faces by computing the colors as a gradient between the vertexes of the faces. This gives the objects a smooth appearance. For the smoothest option, the Per-Pixel Lighting setting needs to be turned on in the Manual Performance Tuning Dialog Box. The colors are computed for individual pixels and gives a smoother appearance. If not, the smooth setting is used instead.

Highlights

The size of the highlights on an object affects the perception of shininess (below). A smaller, more intense highlight makes objects look shinier. The highlight intensity that is set in a visual style does not apply to objects with materials attached.

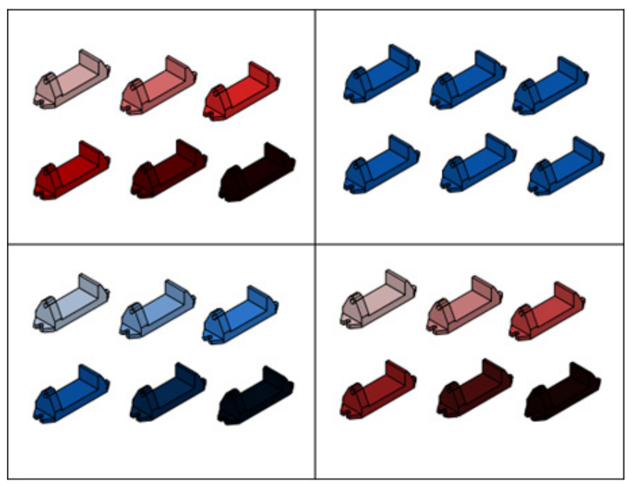


Opacity

The opacity property controls the transparency of an object.

Face Color Modes

Color can be displayed in the normal way, or you can change the face color mode. Monochrome displays all faces in the same color and shaded. Tint uses the same color to shade all faces by changing the hue and saturation values of the color. Desaturate mode softens colors.



To change the display of faces from smooth to faceted

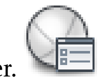
1 Click View tab ➤ 3D Palettes panel ➤ Visual Styles Manager. 

2 With the current visual style selected in the sample images, in Face Settings, Lighting Quality, change Smooth to Faceted.

Command entry: VISUALSTYLES

Toolbar: Visual Styles 

To control the face style

1 Click View tab ➤ 3D Palettes panel ➤ Visual Styles Manager. 

2 With the current visual style selected in the sample images, in Face Settings, Face Style, select one of the following options:

- 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.

Command entry: VISUALSTYLES

Toolbar: Visual Styles 

To control the display of colors on faces

1 Click View tab ➤ 3D Palettes panel ➤ Visual Styles Manager. 

2 With the current visual style selected in the sample images, in Face Settings, Materials and Color, Face Color Mode, select one of the following options:

- **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.

3 If Monochrome or Tint is selected, specify a color. Select Color opens the Select Color dialog box.

Command entry: VISUALSTYLES

Toolbar: Visual Styles 

To make all faces in a viewport transparent

1 If the X-Ray Effect is not turned on: Click Render tab ➤ Visual Styles panel ➤ X-Ray Effect.

2 On the Visual Styles panel, set the opacity by dragging the opacity slider.

Command entry: VSFACEOPACITY

Quick Reference

Commands

VISUALSTYLES

Creates and modifies visual styles and applies a visual style to a viewport.

System Variables

VISUALSTYLESCLOSE

Closes the Visual Styles Manager.

VSFACECOLORMODE

Controls how the color of faces is calculated.

VSFACEHIGHLIGHT

Controls the level of transparency for 3D objects.

VSFACEOPACITY

Turns on and off a preset level of transparency for 3D objects.

VSFACESTYLE

Controls how faces are displayed in the current viewport.

VSLIGHTINGQUALITY

Sets the lighting quality in the current viewport.

VSMONOCOLOR

Sets the color for monochrome and tint display of faces in the visual style applied to the current viewport.

Display Backgrounds and Shadows

The visual style also controls the display of backgrounds and shadows in the viewport.

Backgrounds

You can use a color, a gradient fill, an image, or the sun & sky as a background in the viewport in any 3D visual style, even one that does not shade objects. To use a background, you first create a named view with a background and set the named view as current in the viewport. When Background is set to On in the current visual style, the background is displayed.

Shadows

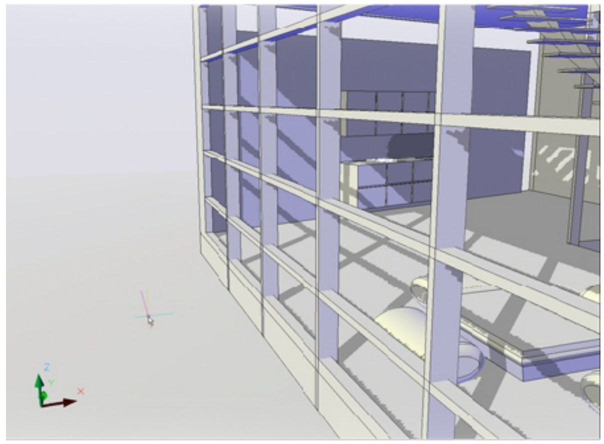
Shaded objects in a viewport can display shadows. Ground shadows are shadows that objects cast on the ground. Full shadows are shadows cast by objects onto other objects. The lighting in the viewport must be from user-created lights or the sun for full shadows to be displayed. Where shadows overlap, they appear darker.

NOTE To display full shadows, hardware acceleration is required. When Enhanced 3D Performance 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.)

Displaying shadows can slow performance. You can turn off shadows in the current visual style while you work and turn them back on when you need them.

In the Properties palette, you can set the Shadow Display property for an object: casts shadows, receives shadows, casts and receives shadows, or ignores shadows.

More options are available for shadows used in rendering.



See also:

- Use Shadows in Rendering

To control the display of shadows in a viewport

1. Click View tab ➤ 3D Palettes panel ➤ Visual Styles Manager. 
2. With the current visual style selected in the sample images, in Environment Settings, Shadow Display, select Off, Full Shadows, or Ground Shadow.

Toolbar: Visual Styles 
Command entry: VISUALSTYLES

To set a solid as a viewport background

1. Click View tab ➤ Views panel ➤ Named Views. 

2 In the View Manager, click the view name you want to change. If the view isn't already displayed, expand the appropriate View list, and then click a view name.

3 In the Properties panel, click the Background Override field.

4 Select Solid from the Background Override drop-down list.

5 In the Background dialog box, click the swatch under the Color section.

6 In the Select Color dialog box, specify the color you want to use for the background.

7 Click OK.

8 In the Background dialog box, click OK.

9 In the View Manager, click Set Current.

10 Click OK.

Command entry: VIEW

Toolbar: View

To set a gradient as a viewport background

1 Click View tab ➤ Views panel ➤ Named Views.

2 In the View Manager, click the view name you want to change. If the view isn't already displayed, expand the appropriate View list, and then click a view name.

3 In the Properties panel, click the Background Override field.

4 Select Gradient from the Background Override drop-down list.

5 In the Background dialog box, specify the desired options for the gradient fill.

- Select Three Color to use a two or three color gradient fill.
- Click on the color swatches to display the Select Color dialog box to specify the colors you want to use for the gradient fill.
- Enter a rotation value for the gradient fill in the Rotation text box.

6 In the Background dialog box, click OK.

7 In the View Manager, click Set Current.

8 Click OK.

Command entry: VIEW

Toolbar: View 

To set an image as a viewport background

1 Click View tab ➤ Views panel ➤ Named Views. 

2 In the View Manager, click the view name you want to change. If the view isn't already displayed, expand the appropriate View list, and then click a view name.

3 In the Properties panel, click the Background Override field.

4 Select Image from the Background Override drop-down list.

5 In the Background dialog box, click Browse and specify the image for the background.

6 Click Adjust Image.

7 In the Adjust Background Image dialog box, specify the desired options for the image.

- Specify the position for the image by choosing an option from the Image Position drop-down list.
- Click Offset or Scale to adjust what functionality the sliders have.
- Click Maintain Aspect Ratio When Scaling to have the image scale uniformily.
- Drag the sliders to adjust the offset or scale of the image based on the current setting.

8 In the Adjust Background Image dialog box, click OK.

9 In the Background dialog box, click OK.

10 In the View Manager, click Set Current.

11 Click OK.

Toolbar: View 
Command entry: VIEW

To set a Sun/Sky as a viewport background

1 Before changing the background, make sure that photometric lighting is enabled:
   - Click Render tab ➤ Lights panel ➤ American Lighting Units
   - Click Render tab ➤ Lights panel ➤ International Lighting Units

2 Click View tab ➤ Views panel ➤ Named Views. 

3 In the View Manager, click the view name you want to change. If the view isn't already displayed, expand the appropriate View list, and then click a view name.

4 In the Properties panel, click the Background Override field.

5 In the Background Override drop-down list, select Sun&Sky.

6 In the Adjust Sun & Sky Background dialog box, specify the desired options for the background.

7 Click OK.

8 In the View Manager, click Set Current. Click OK.

Toolbar: View 
Command entry: LIGHTINGUNITS, VIEW

Quick Reference

Commands

VIEW

VISUALSTYLES

Creates and modifies visual styles and applies a visual style to a viewport.

System Variables

CSHADOW

Sets the shadow display property for a 3D object.

LIGHTINGUNITS

Controls whether generic or photometric lights are used, and indicates the current lighting units.

SHADOWPLANELOCATION

Controls the location of an invisible ground plane used to display shadows.

VSBACKGROUNDS

Controls whether backgrounds are displayed in the visual style applied to the current viewport.

VSSHADOWS

Controls whether a visual style displays shadows.

VISUALSTYLESCLOSE

Closes the Visual Styles Manager.

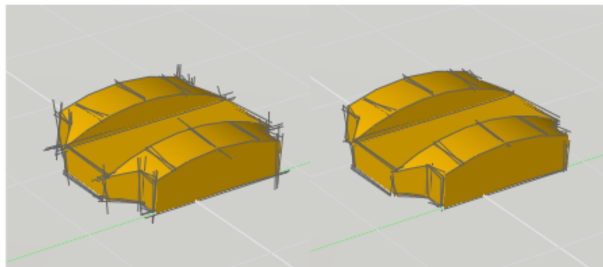
Control the Display of Edges

Edges of different types can be displayed using different colors and linetypes. You can also add special effects such as jitter and overhang to edges.

In a shaded or a wireframe model, the visual style sets the visibility and appearance of isolines, facet edges, silhouette edges, obscured edges, and intersection edges. Facet edges (the edges between planar faces representing a surface) are displayed only when the angle between the facets is smaller than the crease angle value you specify.

Edge modifiers such as Overhang and Jitter produce the look of a model that is still in the conceptual phase. Jitter makes lines look as though they were sketched with a pencil. Overhang produces another kind of hand-drawn effect.

NOTE Plot styles are not available for objects with the Jitter edge modifier applied.



Overhang edges Jitter edges

To control the display of edges and edge colors

1. Click View tab ➤ 3D Palettes panel ➤ Visual Styles Manager. 
2. With the current visual style selected in the sample images, in Edge Settings, Edge Mode, select Facet edges, Isolines, or None.
3. If Isolines is selected, specify the number of lines.
4. In Color, specify a color. Select Color opens the Select Color dialog box.

Command entry: VISUALSTYLES

Toolbar: Visual Styles 

To add an overhang effect to edges

1. Click View tab ➤ 3D Palettes panel ➤ Visual Styles Manager. 
2. With the current visual style selected in the sample images, in Edge Modifiers, click the Overhanging Edges button and adjust the amount of overhang in Overhang.

NOTE The overhang is scaled to look appropriate in viewports of different sizes. The overhang effect is not applied to a line that is less than twice as long as the overhang would be.

Command entry: VISUALSTYLES

Toolbar: Visual Styles

To add a jitter effect to edges

1 Click View tab ➤ 3D Palettes panel ➤ Visual Styles Manager.

2 With the current visual style selected in the sample images, in Edge Modifiers, click the Jitter Edges button and adjust the amount of jitter in Jitter.

Command entry: VISUALSTYLES

Toolbar: Visual Styles

To control display of silhouette edges

1 Click View tab ➤ 3D Palettes panel ➤ Visual Styles Manager.

2 With the current visual style selected in the sample images, in Fast Silhouette Edges, Visible, select Yes or No.

3 If Yes is selected, specify a width for silhouette edges.

Toolbar: Visual Styles
Command entry: VISUALSTYLES

Quick Reference

Commands

VISUALSTYLES

Creates and modifies visual styles and applies a visual style to a viewport.

System Variables

FACETRES

Adjusts the smoothness of shaded and rendered objects and objects with hidden lines removed.

INTERSECTIONCOLOR

Controls the color of polylines at the intersection of 3D surfaces when the visual style is set to 2D Wireframe.

INTERSECTIONDISPLAY

Specifies the display of intersection polylines.

VISUALSTYLESCLOSE

Closes the Visual Styles Manager.

VSEDGECOLOR

Sets the color of edges in the visual style in the current viewport.

VSEDGEJITTER

Makes edges on 3D objects appear wavy, as though they were sketched with a pencil.

VSEDGEOVERHANG

Makes edges on 3D objects extend beyond their intersection for a hand-drawn effect.

VSEDGES

Controls the types of edges that are displayed in the viewport.

VSEDGESMOOTH

Specifies the angle at which crease edges are displayed.

VSHALOGAP

Sets the halo gap in the visual style applied to the current viewport.

VSINTERSECTIONEDGES

Controls the display of intersection edges in the visual style applied to the current viewport.

VSINTERSECTIONCOLOR

Specifies the color of intersection polylines in the visual style applied to the current viewport.

VSINTERSECTIONLTYPE

Sets the linetype for intersection lines in the visual style applied to the current viewport.

VSOBSCUREDEDGES

Controls whether obscured (hidden) edges are displayed.

VSOBSCUREDCOLOR

Specifies the color of obscured (hidden) lines in the visual style applied to the current viewport.

VSOBSCUREDLTYPE

Specifies the linetype of obscured (hidden) lines in the visual style applied to the current viewport.

VSSILHEDGES

Controls display of silhouette edges of solid objects in the visual style applied to the current viewport.

VSSILHWIDTH

Specifies the width in pixels of silhouette edges in the current viewport.

Control Performance

3D graphics display and memory allocation can slow performance on your system. Adaptive degradation, performance tuning and memory tuning are different approaches to delivering the best performance possible.

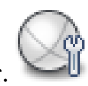
Adaptive Degradation

Adaptive degradation is a way to control performance of features as you use them.

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. Each visual style sets a degradation order appropriate for itself, based on a master order that you can adjust manually.

You set the performance level in frames per second in the Adaptive Degradation and Performance Tuning dialog box. You can also control the order in which features are turned off when performance slows. For example, if you're working on your model with a visual style that has shadows and a background, you can move Shadows and Backgrounds to the top of the list so that they degrade first and preserve edge effects that you need while drafting.

To set the performance level for adaptive degradation

1 Click Render tab ➤ Visual Styles panel ➤ Performance Tuner. 

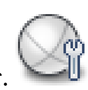
2 In the Adaptive Degradation and Performance Tuning dialog box, check Adaptive Degradation.

3 Enter or select a value for the frames per second. Degradation of the display begins when performance is below this value.

4 Click OK.

Command entry: 3DCONFIG

To change the adaptive degradation order

1 Click Render tab ➤ Visual Styles panel ➤ Performance Tuner. 

2 In the Adaptive Degradation and Performance Tuning dialog box, select a feature in the list.

3 Click Move Up or Move Down to change its order in the list.

   Items at the top of the list degrade first. Click OK.

Command entry: 3DCONFIG

Quick Reference

Commands

3DCONFIG

Sets options that affect 3D display performance.

System Variables

VSBACKGROUNDS

Controls whether backgrounds are displayed in the visual style applied to the current viewport.

VSFACEOPACITY

Turns on and off a preset level of transparency for 3D objects.

VSINTERSECTIONEDGES

Controls the display of intersection edges in the visual style applied to the current viewport.

VSSHADOWS

Controls whether a visual style displays shadows.

VSSILHEDGES

Controls display of silhouette edges of solid objects in the visual style applied to the current viewport.

Performance Tuning

Examines your graphics card and 3D display driver and decides whether to use software or hardware implementation for features that support both.

Features that cannot work properly on your system are turned off. Some features may work but not be recommended for use with your graphics card or 3D graphics display driver. You can turn these on at your own risk.

NOTE If you are using a graphics card that does not support all of the available hardware effects, you can plot a drawing through software emulation. (To enable the emulation of hardware effects in software, enter **3dconfig** at the Command prompt. In the Adaptive Degradation and Performance Tuning dialog box, click Manual Tune. Then click Emulate Unsupported Hardware Effects in Software When Plotting.)

The performance tuner creates a log that reports the graphics card and 3D display driver found on your system and the on or off status of effects. The Current Effects Status section displays the settings of the current display driver. An example of the Current Effects Status section might look like the following:

```
Enhanced 3D Performance:  Available and on
Gooch shader:             Available and using hardware
Full-shadow display:      Available and on
Per-pixel lighting :      Available and on
Texture compression:      Available and on
```

You can manually turn on and off features that your system supports, recommended or not.

The log that is created by the performance tuner not only displays information about the current application driver, but also displays information about all the available application drivers and which effects are supported. This information is in the Available Application Drivers and Effect Support section. By using the information that is displayed, you can make the best decision about which application driver to use for your graphics card or the Software driver. The Available Application Drivers and Effect Support section might look like the following:

```
Software driver
Effect support:
Enhanced 3D Performance:  Not applicable
Gooch shader:             Software emulation only
Per-pixel lighting :      Available and on
Full-shadow display:      Not applicable
Texture compression:      Not applicable
```

```
Autodesk driver:    Certified
Effect support:
Enhanced 3D Performance:  Available
Gooch shader:             Available
Per-pixel lighting :      Available and on
Full-shadow display:      Available
Texture compression:      Available
```

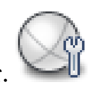
NOTE To display full shadows in viewports and to use Per-pixel lighting, hardware acceleration is required. When Enhanced 3D Performance is off, full shadows cannot be displayed in viewports. (To access these settings, enter **3dconfig** at the Command prompt. In the Adaptive Degradation and Performance Tuning dialog box, click Manual Tune.)

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, 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. Also, there is a reduction in the quality of the images when they are displayed in the viewport or plotted. (To access these settings, enter **3dconfig** at the Command prompt. In the Adaptive Degradation and Performance Tuning dialog box, click Manual Tune.)

Certification Updates

As more graphics cards and 3D display drivers are tested, they are added to a list on the Autodesk website. You can check for updates and download the latest list at any time. In the Adaptive Degradation and Performance Tuning dialog box, click Check for Updates.

To view the performance tuner log

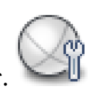
1 Click Render tab ➤ Visual Styles panel ➤ Performance Tuner. 

2 In the Adaptive Degradation and Performance Tuning dialog box, click View Tune Log.

3 Click OK to exit each dialog box.

Command entry: 3DCONFIG

To tune performance manually

1 Click Render tab ➤ Visual Styles panel ➤ Performance Tuner. 

2 In the Adaptive Degradation and Performance Tuning dialog box, click Manual Tune.

3 In the Manual Performance Tuning dialog box, you can enable hardware acceleration if the icon next to the check box is a green check mark or a yellow warning sign.

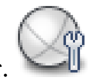
4 In General Settings, you can clear or turn down settings to improve performance.

5 (Optional) Use the Reset to Recommended Values option.

6 Click OK to exit each dialog box.

Command entry: 3dconfig

To check for graphics card and driver certification updates

1 Click Render tab ➤ Visual Styles panel ➤ Performance Tuner. 

2 In the Adaptive Degradation and Performance Tuning dialog box, click Check for Updates.

  A web page is displayed that describes the certification program.

3 Look for your graphics card and driver in the list. If you find them, follow the download instructions.

4 Click OK to exit each dialog box.

Command entry: 3DCONFIG

Quick Reference

Commands

3DCONFIG

Sets options that affect 3D display performance.

System Variables

VSSHADOWS

Controls whether a visual style displays shadows.

Memory Tuning

Performance can also be improved by adding memory to your system. This is especially true when working on larger models.

The system requirements for AutoCAD require at least 512 MB of physical memory (RAM) for working in 2D. For creating and working with 3D models, at least 2 GB of RAM is required.

The size and complexity of a model often defines how efficiently an application runs. If you notice increased hard drive activity, it means that physical memory has been exceeded and data is being passed to a swap file (virtual memory).

A swap file is an area on the hard drive that Windows uses as if it were physical memory (RAM). The swap file size is basically a limit which restricts the total virtual size of the AutoCAD process. A good rule of thumb for configuring your swap file is three times the amount of physical memory on your system. This usually sets the limit high enough that AutoCAD doesn't run out of swap space.

Typically, systems are limited to a maximum of 2 GB of virtual memory. On some systems, you can set the **/3GB** switch that allows applications to use up to 3 GB of virtual memory. Before attempting to set the switch, you need to find out the following:

- If your system supports the switch.
- If other device drivers (video card, plotter drivers, etc.) may be affected if the switch is set.
- How to set the switch.

Refer to the Microsoft website for details regarding the **/3GB** switch.

To check physical memory (RAM)

1. On the Start menu (Windows), click Control Panel (or Settings ➤ Control Panel).
2. In the Control Panel, click System. You might have to click Performance and Maintenance and then System.
3. On the General tab, review the information about your computer. The amount of RAM should be listed on this tab.

To check swap file allocation

1. On the Start menu (Windows), click Control Panel (or Settings ➤ Control Panel).
2. In the Control Panel, click System. You might have to click Performance and Maintenance and then System.
3. Open the Advanced tab and click the Settings (or Performance Options) button under Performance.

4 Check the allocated size under the Virtual Memory group, the file size will be listed after the text “Total Paging File Size For All Drives”. You might have to click the Advanced tab first in order to see the allocated size.

Use Viewing Tools

10

When working in 3D, you'll often want to display different views so that you can see and verify the 3D effects in your drawing.

Specify 3D Views

You can control the 3D navigation display, projection, and visualization tools.

Overview of 3D Views

You can create an interactive view of your drawing in the current viewport.

Using the 3D viewing and navigation tools, you can navigate through a drawing, set up a camera for a specific view, and create animations to share your design with others. You can orbit, swivel, walk, and fly around a 3D model, set up a camera, create a preview animation, and record motion path animations that you can distribute to others to visually convey the intent of your design.

Quick Reference

Commands

3DCLIP

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

3DCORBIT

Rotates the view in 3D space with continuous motion.

3DDISTANCE

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

3DFLY

Changes the 3D view in a drawing interactively to create the appearance of flying through the model.

3DFORBIT

Rotates the view in 3D space without constraining roll.

3DORBIT

Rotates the view in 3D space, but constrained to horizontal and vertical orbit only.

3DORBITCTR

Sets the center of rotation in 3D Orbit view.

3DPAN

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

3DSWIVEL

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

3DWALK

Changes the 3D view in a drawing interactively to create the appearance of walking through the model.

3DZOOM

Zooms in and out in a perspective view.

ANIPATH

Saves an animation file of a camera moving or panning in a 3D model.

CAMERA

Sets a camera and target location to create and save a 3D perspective view of objects.

VIEW

WALKFLYSETTINGS

Controls the walk and fly navigation settings.

System Variables

CAMERADISPLAY

Turns the display of camera objects on or off.

CAMERAHEIGHT

Specifies the default height for new camera objects.

Use 3D Navigation Tools

3D navigation tools allow you to view objects in a drawing from different angles, heights, and distances.

Use the following 3D tools to orbit, swivel, adjust distance, zoom, and pan in a 3D view.

- **3D Orbit**. Moves around a target. The target of the view stays stationary while the camera location, or point of view, moves. The center of the viewport, not the center of the objects you're viewing, is the target point.
- **Constrained Orbit**. Constrains 3D Orbit along the *XY* plane or the *Z* axis. (3DORBIT)
- **Free Orbit**. Orbits in any direction without reference to the planes. The point of view is not constrained along the *XY* plane of the *Z* axis. (3DFORBIT)
- **Continuous Orbit**. Orbits continuously. Click and drag in the direction you want the continuous orbit to move, and then release the mouse button. The orbit continues to move in that direction. (3DCORBIT)
- **Adjust Distance**. Changes the distance of objects as you move the cursor vertically. You can make objects appear larger or smaller, and you can adjust the distance. (3DDISTANCE)
- **Swivel**. 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. (3DSWIVEL)
- **Zoom**. Simulates moving the camera closer to an object or farther away. Zooming in magnifies the image. (3DZOOM)
- **Pan**. Starts the interactive 3D view and enables you to drag the view horizontally and vertically. (3DPAN)

Animate a Navigation

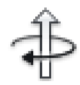
You can create a preview animation of any navigation. Create the preview to fine-tune your animation before you create a motion path animation. You can create, record, play back, and save the animation. For more information about previewing an animation, see Create Preview Animations on page 403. For more information about creating motion path animations, see Create Motion Path Animations on page 406.

To start 3D Orbit view

1 Select one or more objects that you want to view with 3DORBIT or select no objects if you want to view the entire drawing.

   NOTE OLE objects and raster objects do not appear in the 3D Orbit view.

   ---

2 Click View tab ➤ Navigate panel ➤ Orbit. 

3 Use one of the following methods to orbit around the objects:

   - To rotate along the *XY* plane, click in the drawing and drag the cursor left or right.
   - To rotate along the *Z* axis, click in the drawing and drag the cursor up or down.
   - To allow unconstrained orbiting along the *XY* plane and the *Z* axis, press Shift while you drag the cursor. An arcball is displayed and you can use the 3D Free Orbit (3DFORBIT) interaction.

4 Press Enter.

Command entry: 3DORBIT

Toolbar: 3D Navigation 

To switch to a different 3D navigation mode

1 Start any 3D navigation command.

2 Right-click in the drawing area. Click Other Navigation Modes, and click another navigation mode.

To choose a parallel or perspective projection in a 3D view

1. Start any 3D navigation command.
2. Right-click in the drawing area. Click Parallel or Perspective.
   A check mark is displayed next to the current projection.

To change a visual style in a 3D view

1. Start any 3D navigation command, and right-click in the drawing area. Click Visual Styles.
2. Select a visual style.

For more information about visual styles, see Use a Visual Style to Display Your Model on page 293.

To start a continuous orbit

1. Click View tab ➤ Navigate panel ➤ Continuous Orbit.
2. Click in the drawing and drag the cursor to start a continuous motion. When you release cursor, the orbit continues in the direction that you were dragging.

Command entry: 3DCORBIT

Toolbar: Orbit

To reset a view

- Start any 3D navigation command, and right-click in the drawing area. Click Reset View.

The view that was originally displayed when you entered the 3D navigation mode is restored.

To use preset views

1. Start any 3D navigation command, and right-click in the drawing area. Click Preset Views.
2. Click a view.

To display a named view

1 Start any 3D navigation command, and right-click in the drawing area. Click Named Views.

2 Click a named view or a camera.

Quick Reference

Commands

3DCORBIT

Rotates the view in 3D space with continuous motion.

3DDISTANCE

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

3DFORBIT

Rotates the view in 3D space without constraining roll.

3DORBIT

Rotates the view in 3D space, but constrained to horizontal and vertical orbit only.

3DORBITCTR

Sets the center of rotation in 3D Orbit view.

3DPAN

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

3DSWIVEL

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

3DZOOM

Zooms in and out in a perspective view.

System Variables

PERSPECTIVE

Specifies whether the current viewport displays a perspective view.

PERSPECTIVECLIP

Determines the location of eyepoint clipping.

Walk and Fly Through a Drawing

You can simulate walking and flying through a 3D drawing.

When you walk through a model, you travel along the XY plane. When you fly through a model, you are not constrained by the XY plane, so you appear to “fly” over an area in a model.

Use the Keyboard and Mouse Interactions to Walk and Fly

You can use a standard set of keys and mouse interactions to walk and fly through a drawing. Use the four arrow keys or the W, A, S, and D keys to move up, down, left, or right. To toggle between walk and fly mode, press the F key. To specify the direction of the view, drag the mouse in the direction you want to look.

NOTE The Walk and Fly Navigation Mappings balloon provides information about the keyboard and mouse actions that control walk and fly modes. The appearance of the balloon depends on the display option selected in the Walk and Fly settings dialog box.

Display a Top View of a Model as You Walk or Fly

You can keep track of your position in a 3D model as you walk or fly through it. When you start 3DWALK or 3DFLY, the Position Locator window displays a top view of the model. A position indicator shows your location in relationship to the model, and a target indicator shows the model you are walking or flying through. You can edit the position settings in the Position Locator window before you start walk or fly mode, or while you are moving in the model.

NOTE If performance slows when the Position Locator window is displayed, you can close the window.

Specify Walk and Fly Settings

Specify walk and fly settings in the 3D Navigate panel on the the ribbon on page 22or in the Walk and Fly Settings dialog box. You can set the default step size, the number of steps per second, and other display settings.

Animate Walk and Fly Navigation

You can create a preview animation of any navigation, including walking and flying through a drawing. Create the preview to fine-tune your animation before you create a motion path animation. You can create, record, play back, and save the animation. For more information about previewing an animation, see Create Preview Animations on page 403. For more information about creating motion path animations, see Create Motion Path Animations on page 406.

To start walk mode

- Click View menu ➤ Walk and Fly ➤ Walk.

Command entry: 3DWALK

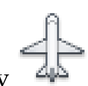
Toolbar: 3D Navigation, Walk and Fly 

Shortcut menu: Start any 3D navigation command, right-click in the drawing area, and select Other Navigation Modes ➤ Walk.

To start fly mode

- Click View menu ➤ Walk and Fly ➤ Fly.

Command entry: 3DFLY

Toolbar: 3D Navigation, Walk and Fly 

Shortcut menu: Start any 3D navigation command, right-click in the drawing area, and select Other Navigation Modes ➤ Fly.

To specify Walk or Fly settings

1. Click View menu ➤ Walk and Fly ➤ Walk and Fly Settings.
2. In the Walk and Fly Settings dialog box, under Settings, click the button next to a display option.
3. In the Display Position Locator Window option, clear the check box if you do not want to display the window.

NOTE If performance slows when this window is displayed, you may want to close the Position Locator window.

4 In the Current Drawing Settings section, under Walk/Fly Step Size, enter a number to set the step size in drawing units.

5 In the Steps Per Second option, enter a number between 1 and 30.

6 Click OK.

Command entry: WALKFLYSETTINGS

Toolbar: 3D Navigation, Walk and Fly 

To change the viewing position in the Position Locator window

1 Do one of the following:
   - Click View menu ➤ Walk and Fly ➤ Walk.
   - Click View menu ➤ Walk and Fly ➤ Fly.

2 In the Position Locator window, Preview area, click the position indicator (a colored dot) and drag it to a new position.

3 If the target indicator is displayed, click it and drag it to the new target.

4 In the General section, make any changes to the current settings.

5 Continue walking or flying through the model.

Toolbar: Walk and Fly
Command entry: 3DWALK, 3DFLY

To turn off the Position Locator window

1 Click View menu ➤ Walk and Fly ➤ Walk and Fly Settings.

2 In the Walk and Fly Settings dialog box, clear the check mark next to Display Position Locator. Click OK.

Command entry: WALKFLYSETTINGS

Toolbar: 3D Navigation, Walk and Fly 
Shortcut menu: While in any 3D Navigation mode, right-click. Click Walk And Fly Settings.

Quick Reference

Commands

3DFLY

Changes the 3D view in a drawing interactively to create the appearance of flying through the model.

3DWALK

Changes the 3D view in a drawing interactively to create the appearance of walking through the model.

WALKFLYSETTINGS

Controls the walk and fly navigation settings.

Create a 3D Dynamic View (DVIEW)

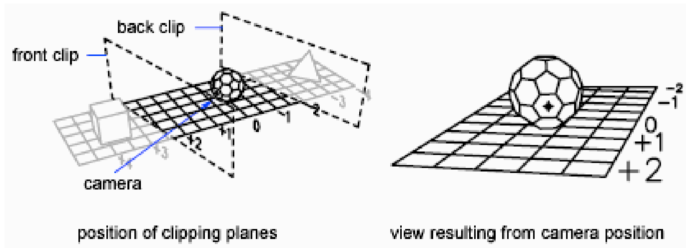
You can change a view without interrupting your current operation using a feature that combines panning and zooming.

With dynamic viewing, you can display the effects of changing your viewpoint as you make the changes. Using this method, you can also simplify your view temporarily by choosing only the objects that you need to determine the view. Alternatively, if you press Enter without selecting any objects, 3D Dynamic View displays a model of a small house instead of your actual drawing. You can use this house to define the viewing angle and distance. When your adjustments are complete and you exit the command, the changes are applied to the entire 3D model in the current view.

NOTE More powerful options for dynamic viewing in 3D are available in the 3DORBIT command. For more information, see Use 3D Navigation Tools on page 323.

Set Clipping Planes

You can create cutaway, or section, views of your drawing by positioning front and back clipping planes that control the visibility of objects based on their distance from a theoretical camera. You can move the clipping planes perpendicular to the line of sight between the camera and target (where the camera is pointing). Clipping removes the display of objects from the front and back of clipping planes. The following illustration shows how clipping planes work:



NOTE You can also set clipping planes when you create a camera glyph. For more information, see Change Camera Properties on page 398.

To set clipping planes (DVIEW)

1. At the Command prompt, enter **dview**.
2. Select the objects on which to base the view.
3. At the Command prompt, enter **cl** (Clip).
4. Enter **f** to set a front clipping plane or **b** to set a back clipping plane, or press Enter.
5. Position the clipping plane by dragging the slider or entering a distance from the target.
6. Press Enter to exit the command.

   NOTE You can also set clipping planes when you create a camera glyph. For more information, see Change Camera Properties on page 398.

Command entry: DVIEW

To display the default view and UCS

- Click View menu ➤ 3D Views ➤ Plan View ➤ World UCS.

Command entry: PLAN

Quick Reference

Commands

DVIEW

Defines parallel projection or perspective views by using a camera and target.

PLAN

Displays the plan view of a specified user coordinate system

System Variables

VIEWDIR

Stores the viewing direction in the current viewport, expressed in UCS coordinates.

VIEWTWIST

Stores the view rotation angle for the current viewport measured relative to the WCS.

WORLDVIEW

Use ViewCube

ViewCube provides visual feedback of the current orientation of a model. You can use the ViewCube tool to adjust the viewpoint of the model.

Overview of ViewCube

ViewCube is a 3D navigation tool that is displayed when you are working in a 3D visual style. With ViewCube, you can switch between standard and isometric views.

The ViewCube tool is a persistent, clickable and draggable interface that you use to switch between standard and isometric views of your model. When you display the ViewCube tool, it is shown in one of the corners of the window over the model in an inactive state. While the ViewCube tool is inactive, it provides visual feedback about the current viewpoint of the model as view changes occur. When the cursor is positioned over the ViewCube tool, it becomes active; you can switch to one of the available preset views, roll the current view, or change to the Home view of the model.

Edge Corner Face

Control the Appearance of ViewCube

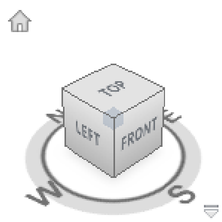
The ViewCube tool is displayed in one of two states: inactive and active. When the ViewCube tool is inactive, it appears partially transparent by default so that it does not obscure the view of the model. When active, it is opaque and may obscure the view of the objects in the current view of the model.

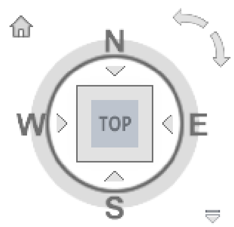
In addition to controlling the inactive opacity level of the ViewCube tool, you can also control the following properties for the ViewCube tool:

- Size
- Position
- Display of the UCS menu
- Default orientation
- Compass display

Using the Compass

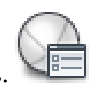
The compass is displayed below the ViewCube tool and indicates which direction North is defined for the model. You can click a cardinal direction letter on the compass to rotate the model, or you can click and drag one of the cardinal direction letters or the compass ring to interactively rotate the model around the center of the view.





To display or hide the ViewCube tool

1 Click View tab ➤ Views panel ➤ ViewCube. 

2 Click View tab ➤ 3D Palettes panel ➤ Visual Styles. 

3 Click in the viewport to make it current.

4 In the Visual Styles Manager, double-click one of the sample images of the available visual styles other than 2D Wireframe.

Command entry: NAVVCUBE, VISUALSTYLES

To control the position of the ViewCube tool

1 Click View tab ➤ Views panel ➤ ViewCube. 

2 Right-click the ViewCube tool and click ViewCube Settings.

3 In the ViewCube Settings dialog box, under Display, select one of the available positions from the On-screen Position drop-down list.

4 Click OK.

Command entry: NAVVCUBE

To control the size of the ViewCube tool

1 Click View tab ➤ Views panel ➤ ViewCube. 
2 Right-click the ViewCube tool and click ViewCube Settings.
3 In the ViewCube Settings dialog box, Display, ViewCube Size, select or clear Automatic. If Automatic is cleared, drag the ViewCube Size slider left or right.

   Dragging the slider to the left decreases the size of the ViewCube tool, while dragging the slider to the right increases the size of the ViewCube tool.
4 Click OK.

Command entry: NAVVCUBE

To control the inactive opacity of the ViewCube tool

1 Click View tab ➤ Views panel ➤ ViewCube. 
2 Right-click the ViewCube tool and click ViewCube Settings.
3 In the ViewCube Settings dialog box, under Display, drag the Inactive Opacity slider left or right.

   Dragging the slider to the left increases the transparency of the ViewCube tool, while dragging the slider to the right increases the opacity of the ViewCube tool.
4 Click OK.

Command entry: NAVVCUBE

To display the compass for the ViewCube tool

1 Click View tab ➤ Views panel ➤ ViewCube. 
2 Right-click the ViewCube tool and click ViewCube Settings.
3 In the ViewCube Settings dialog box, select Show Compass Below the ViewCube.

The compass is displayed below the ViewCube tool and indicates which direction North is in the model.

4 Click OK.

Command entry: NAVVCUBE

Quick Reference

Commands

NAVVCUBE

Controls the visibility and display properties of the ViewCube tool.

OPTIONS

Customizes the program settings.

System Variables

NAVVCUBEDISPLAY

Controls the display of the ViewCube tool in the current viewport when the 3D graphics system is active.

NAVVCUBELOCATION

Identifies the corner in a viewport where the ViewCube tool is displayed.

NAVVCUBEOPACITY

Controls the opacity of the ViewCube tool when inactive.

NAVVCUBESIZE

Specifies the size of the ViewCube tool.

ViewCube Menu

Use the ViewCube menu to restore and define the Home view of a model, switch between view projection modes, and change the interactive behavior and appearance of the ViewCube tool.

The ViewCube menu has the following options:

- **Home.** Restores the Home view saved with the model.

- **Parallel.** Switches the current view to parallel projection.
- **Perspective.** Switches the current view to perspective projection.
- **Perspective with Ortho Faces.** Switches the current view to perspective projection unless the current view aligns with a face view defined on the ViewCube tool.
- **Set Current View as Home.** Defines the Home view of the model based on the current view.
- **ViewCube Settings.** Displays the dialog box where you can adjust the appearance and behavior of the ViewCube tool.
- **Help.** Launches the online Help system and displays the topic for the ViewCube tool.

To display the ViewCube menu

To display the ViewCube menu, do one of the following:

- Right-click on the compass, Home icon, or the main area of the ViewCube tool.
- Click the context menu button located below the ViewCube tool.

Command entry: NAVVCUBE

Quick Reference

Commands

NAVVCUBE

Controls the visibility and display properties of the ViewCube tool.

OPTIONS

Customizes the program settings.

System Variables

PERSPECTIVE

Specifies whether the current viewport displays a perspective view.

Reorient the View of a Model with ViewCube

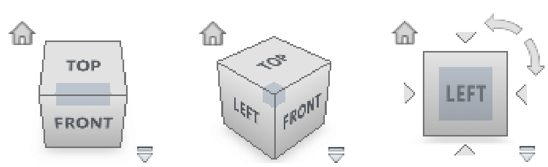
The ViewCube tool offers many intuitive ways to reorient the view of a model.

Reorient the Current View

You can reorient the current view of a model by clicking predefined areas on the ViewCube tool or dragging the ViewCube tool.

The ViewCube tool provides twenty-six defined areas you can click to change the current view of a model. The twenty-six defined areas are categorized into three groups: corner, edge, and face. Of the twenty-six defined areas, six of the areas represent standard orthogonal views of a model: top, bottom, front, back, left, and right. Orthogonal views are set by clicking one of the faces on the ViewCube tool.

You use the other twenty defined areas to access angled views of a model. Clicking one of the corners on the ViewCube tool reorients the current view of the model to a three-quarter view, based on a viewpoint defined by three sides of the model. Clicking one of the edges reorients the view of the model to a half view based on two sides of the model.



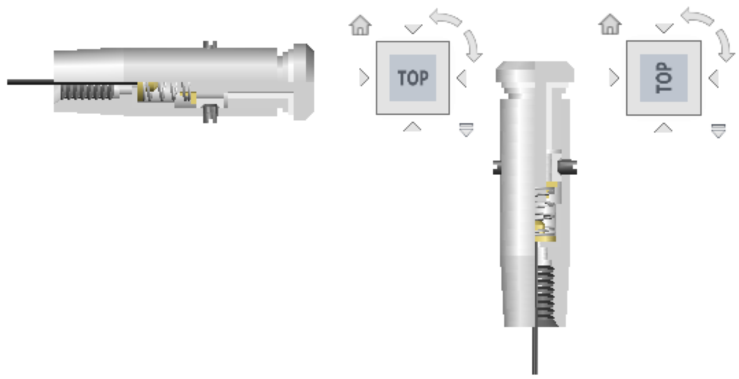
Edge **Corner** **Face**

You can also click and drag the ViewCube tool to reorient the view of a model to a custom viewpoint other than one of the twenty-six predefined viewpoints. As you drag, the cursor changes to indicate that you are reorienting the current view of the model. If you drag the ViewCube tool close to one of the preset orientations and it is set to snap to the closest view, the ViewCube tool rotates to the closest preset orientation.

The outline of the ViewCube tool helps you identify the form of orientation it is in: freeform or constrained. When the ViewCube tool is in freeform orientation, not orientated to one of the twenty-six predefined views, its outline is displayed as dashed. The ViewCube tool is outlined in a solid continuous line when it is constrained to one of the predefined views.

Roll a Face View

When you view a model from one of the face views, two roll arrow buttons are displayed near the ViewCube tool. Use the roll arrows to rotate the current view 90 degrees clockwise or counterclockwise around the center of the view.



Switch to an Adjacent Face

When the ViewCube tool is active while viewing a model from one of the face views, four orthogonal triangles are displayed near the ViewCube tool. You use these triangles to switch to one of the adjacent face views.

To interactively reorient the view

1 Click View tab ➤ Views panel ➤ ViewCube. 

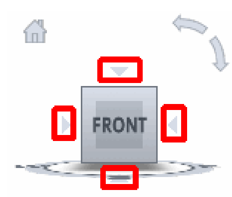
2 Click the ViewCube tool or the compass below the ViewCube tool, hold down the left mouse button, and drag in the direction that you want to orbit the model.

Command entry: NAVVCUBE

To view an adjacent face

NOTE Make sure a face view is current.

- Click one of the triangles displayed near the edges of the ViewCube tool.



Command entry: NAVVCUBE

To interactively reorient the view

- Click the ViewCube tool, hold down the left mouse button, and drag in the direction that you want to orbit the model.

Command entry: NAVVCUBE

To use animated transitions when reorienting a view to a preset orientation

1. Click View tab ➤ Views panel ➤ ViewCube. 
2. Right-click the ViewCube tool, and click ViewCube Settings.
3. In the ViewCube Settings dialog box, under When Clicking on the ViewCube, select Use View Transitions when Switching Views.

   Transitions from one view to another appear animated when you click a predefined area on the ViewCube tool.
4. Click OK.

Command entry: NAVVCUBE

To automatically fit the model after a view orientation

1. Click View tab ➤ Views panel ➤ ViewCube. 

2 Right-click the ViewCube tool, and click ViewCube Settings.

3 In the ViewCube Settings dialog box, under When Clicking on the ViewCube, click Zoom to Extents After View Change.

   Clicking a predefined area of the ViewCube tool reorients the model and fits the model to the window.

4 Click OK.

Command entry: NAVVCUBE

To roll a face view

NOTE Make sure a face view is displayed.

- Click one of the roll arrows displayed above and to the right of the ViewCube tool.
  The left roll arrow rotates the view 90 degrees counterclockwise; the right roll arrow rotates the view 90 degrees clockwise.

Command entry: NAVVCUBE

Quick Reference

Commands

NAVVCUBE

Controls the visibility and display properties of the ViewCube tool.

OPTIONS

Customizes the program settings.

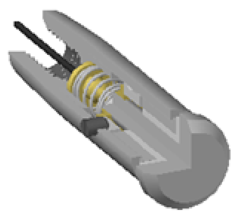
Set the View Projection Mode

View projection produces realistic visual effects of a model.

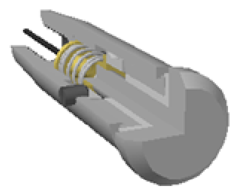
The ViewCube tool supports two different view projections: perspective and orthographic. Orthographic projection is also referred to as parallel projection. Perspective projected views are calculated based on the distance from a theoretical camera and target point. The shorter the distance between the camera and the target point, the more distorted the perspective effect appears; greater distances produce less distorted affects on the model. Orthographic projected views display all the points of a model being projected parallel to the screen.

Orthographic projection mode makes it easier to work with a model due to all the edges of the model appearing as the same size, regardless of the distance from the camera. Orthographic projection mode though, is not how you commonly see objects in the real world. Objects in the real world are seen in perspective projection. So when you want to generate a rendering or hidden line view of a model, using perspective projection will give the model a more realistic look.

The following illustration shows the same model viewed from the same viewing direction, but with different view projections.



Parallel



Perspective

When you change the view for a model, the view is updated using the previous projection mode unless the current projection mode for the ViewCube tool is Perspective with Ortho Faces. The Perspective with Ortho Faces mode forces all views to be displayed in perspective projection unless the model is being viewed from one of the face views: top, bottom, front, back, left, or right.

To change the view projection mode

1. Click View tab ➤ Views panel ➤ ViewCube. 
2. Right-click on the ViewCube tool and click one of the following options:
   - Parallel
   - Perspective
   - Perspective with Ortho Faces

Command entry: NAVVCUBE

Quick Reference

Commands

NAVVCUBE

Controls the visibility and display properties of the ViewCube tool.

OPTIONS

Customizes the program settings.

System Variables

NAVVCUBEDISPLAY

Controls the display of the ViewCube tool in the current viewport when the 3D graphics system is active.

PERSPECTIVE

Specifies whether the current viewport displays a perspective view.

Home View

You can define a Home view for a model so you can restore a familiar view when you use the navigation tools.

The Home view is a special view stored with a model that makes it easy to return to a known or familiar view. You can define any view of the model as the Home view. The saved Home view can be applied to the current view by clicking the Home button above the ViewCube tool or from the ViewCube menu.

When you open a drawing that was created in a release earlier than AutoCAD 2008, the extents of a model are used as the default Home view. Drawings created with AutoCAD 2010 have a Home view defined with a Top/Left/Front orientation. While you can use the Home view to navigate back to a familiar view, you can also use it to generate the thumbnail preview when you save a model instead of using the last saved view.

The Thumbnail Preview Settings dialog box is used to control the thumbnail preview for a drawing when saved (the Home or the last saved view). Along with defining which view is used for the thumbnail preview, you can also set the Home view back to its default view.

To define the Home view

- Right-click the ViewCube tool, and click Set Current View as Home.

Ribbon: View tab ➤ Views panel ➤ ViewCube Display 
Menu: View ➤ Display ➤ ViewCube ➤ On
Command entry: NAVVCUBE

To reorient the model to the Home view

Use one of the following methods:

- Click the Home button () located near the ViewCube tool.
- Right-click the ViewCube tool, and click Go Home.

Ribbon: View tab ➤ Views panel ➤ ViewCube Display 
Menu: View ➤ Display ➤ ViewCube ➤ On
Command entry: NAVVCUBE

To reset the Home view to the default view

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Open and Save tab, under File Save, click Thumbnail Preview Settings.
3. In the Thumbnail Preview Settings dialog box, click Reset Home to Default.
4. Click OK to close the Thumbnail Preview Settings dialog box.
5. Click OK to close the Options dialog box.

Command entry: OPTIONS

Quick Reference

Commands

NAVVCUBE

Controls the visibility and display properties of the ViewCube tool.

OPTIONS

Customizes the program settings.

Examine Individual Objects with ViewCube

You can examine individual objects of a model using the ViewCube tool.

With the ViewCube tool, you can define the center of a view based on one or more selected objects. After you select an object or objects and you use the ViewCube tool is used to change the orientation of the model, the model rotates around the center of the view. The center of the view is calculated by the extents of the selected objects.

To examine an individual object with ViewCube

1. In the model, select one or more objects to define the centerpoint of the view.
2. Click one of the preset locations on the ViewCube tool, or click and drag the ViewCube tool to reorient the view of the model.

   The ViewCube tool reorients the view of the model based on the centerpoint of the selected objects.

Ribbon: View tab ➤ Views panel ➤ ViewCube Display 
Menu: View ➤ Display ➤ ViewCube ➤ On
Command entry: NAVVCUBE

Quick Reference

Commands

NAVVCUBE

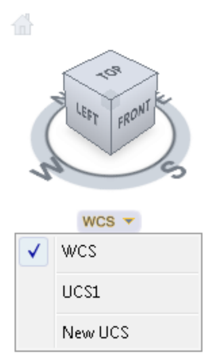
Controls the visibility and display properties of the ViewCube tool.

Change the UCS with ViewCube

With the ViewCube tool you can change the current UCS for the model to one of the named UCSs saved with the model or you can define a new UCS.

The UCS menu, located below the ViewCube tool, displays the name of the current UCS in the model. From the menu, you can restore one of the named UCSs saved with the model, switch to WCS, or define a new UCS. With the WCS item on the menu, you can switch the coordinate system from the current UCS to WCS. With the new UCS, you can rotate the current UCS based on one, two, or three points to define a new UCS. When you click New UCS, a new UCS is defined with the default name of Unnamed. To save the UCS with a name so it can be restored later, use the Named option.

You can orient the ViewCube tool with the current UCS or WCS. By orienting the ViewCube tool with the current UCS, you know in which direction you are modeling. Orienting the ViewCube tool with the WCS, you can navigate the model based on the North and Up directions of the model. The settings for controlling the orientation of the ViewCube tool are in the ViewCube Settings dialog box.



To control the display of the UCS menu

1 Click View tab ➤ Views panel ➤ ViewCube. 
2 Right-click on the ViewCube tool and click ViewCube Settings.
3 In the ViewCube Settings dialog box, under Display, click Show UCS Menu.

   The UCS menu is displayed below the ViewCube tool. You can see which UCS is current and switch to a different named UCS.

4 Click OK.

Command entry: NAVVCUBE

To switch to a different UCS

1 Click View tab ➤ Views panel ➤ ViewCube. 

2 On the UCS menu located below the ViewCube tool, click the down arrow next to the current UCS name.

3 Select a named UCS from the list.

Command entry: NAVVCUBE

To switch to WCS

1 Click View tab ➤ Views panel ➤ ViewCube. 

2 On the UCS menu located below the ViewCube tool, click the down arrow next to the current UCS name.

3 Select a WCS from the list.

Command entry: NAVVCUBE

To set the orientation of the ViewCube tool based on the current UCS or WCS

1 Click View tab ➤ Views panel ➤ ViewCube. 

2 Right-click the ViewCube tool and click ViewCube Settings.

3 In the ViewCube Settings dialog box, select Orient ViewCube to Current UCS.

   The ViewCube tool aligns with the current UCS.

4 Click OK.

Command entry: NAVVCUBE

Quick Reference

Commands

NAVVCUBE

Controls the visibility and display properties of the ViewCube tool.

OPTIONS

Customizes the program settings.

UCS

Manages user coordinate systems.

System Variables

NAVVCUBEORIENT

Controls whether the ViewCube tool reflects the current UCS or WCS.

Navigate with SteeringWheels

SteeringWheels are tracking menus that allow you to access different 2D and 3D navigation tools from a single tool.

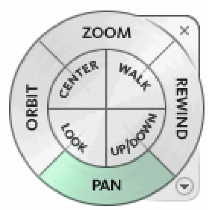
Overview of SteeringWheels

SteeringWheels are tracking menus that are divided into different sections known as wedges. Each wedge on a wheel represents a single navigation tool.

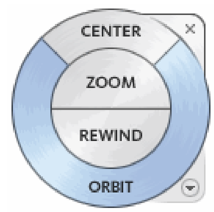
SteeringWheels, also known as wheels, can save you time by combining many of the common navigation tools into a single interface. Wheels are specific to the context from which a model is being viewed.

The following illustrations show the different wheels available:

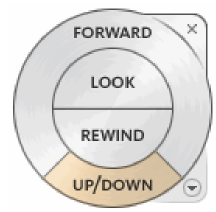
2D Navigation Wheel



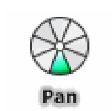
Full Navigation Wheel



View Object Wheel (Basic Wheel)

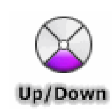


Tour Building Wheel (Basic Wheel)



Mini Full Navigation Wheel

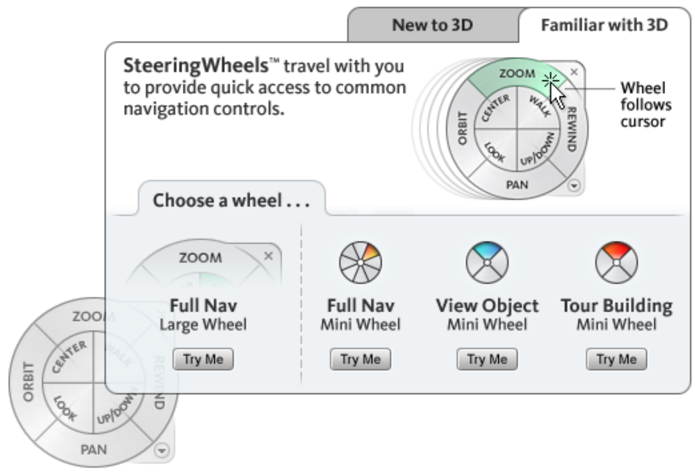
Mini View Object Wheel



Mini Tour Building Wheel

First Contact Balloon

When a SteeringWheel is displayed the first time and a 3D view is current, the First Contact balloon for the wheels is displayed. The First Contact balloon serves as an introduction to the purpose of the wheels and shows how to use them.



Display and Use Wheels

Pressing and dragging on a wedge of a wheel is the primary mode of interaction. After a wheel is displayed, click one of the wedges and hold down the button on the pointing device to activate the navigation tool. Drag to reorient the current view. Releasing the button returns you to the wheel.

Appearance of the Wheels

You can control the appearance of the wheels by switching between the different styles of wheels that are available, or by adjusting the size and opacity. Wheels (except the 2D Navigation wheel) are available in two different styles: big and mini.

The size of a wheel controls how large or small the wedges and labels appear on the wheel; the opacity level controls the visibility of the objects in the model behind the wheel.

Wheel Tooltips, Tool Messages, and Tool Cursor Text

Tooltips are displayed for each button on a wheel as the cursor is moved over them. The tooltips appear below the wheel and identify what action will be performed if the wedge or button is clicked.

Similar to tooltips, tool messages and cursor text are displayed when you use one of the navigation tools from a wheel. Tool messages are displayed when a navigation tool is active; they provide basic instructions about using the tool. Tool cursor text displays the name of the active navigation tool near the cursor. Disabling tool messages and cursor text only affects the messages that are displayed when using the mini wheels or the big Full Navigation wheel.

To display a wheel

Do one of the following:

- Click View tab ➤ Navigate panel ➤ SteeringWheels flyout ➤ click one of the available SteeringWheels.
- Click View menu ➤ SteeringWheels
- Right-click over the drawing window and click SteeringWheels
- On the status bar, click SteeringWheels

Command entry: NAVSWHEEL

To close a wheel

Use one of the following methods to close the wheel:

- Press Esc or Enter
- Click the Close button
- Right-click on the wheel and click Close Wheel

Command entry: NAVSWHEEL

To change the size of the wheels

1. Click View tab ➤ Navigate panel ➤ SteeringWheels flyout ➤ click one of the available SteeringWheels.
2. Right-click on the wheel and click SteeringWheel Settings.
3. In the SteeringWheels Settings dialog box, under Big Wheels or Mini Wheels, slide the Wheel Size slider left or right.

   Sliding the slider to the left decreases the size of the wheel, while sliding the slider to the right increases the size of the wheel.
4. Click OK.

Menu: View ➤ SteeringWheels
Command entry: NAVSWHEEL

Shortcut menu: Right-click over the drawing window and click SteeringWheels

Toolbar: Status bar ➤ SteeringWheels

To change the opacity of the wheels

1 Click View tab ➤ Navigate panel ➤ SteeringWheels flyout ➤ click one of the available SteeringWheels.

2 Right-click on the wheel and click SteeringWheel Settings.

3 In the SteeringWheels Settings dialog box, under Big Wheels or Mini Wheels, slide the Wheel Opacity slider left or right.

   Sliding the slider to the left increases the transparency of the wheel, while sliding the slider to the right decreases the transparency of the wheel making it more opaque.

4 Click OK.

Command entry: NAVSWHEEL

Menu: View ➤ SteeringWheels

Shortcut menu: Right-click over the drawing window and click SteeringWheels

Toolbar: Status bar ➤ SteeringWheels

To control the startup placement of the wheels

1 Click View tab ➤ Navigate panel ➤ SteeringWheels flyout ➤ click one of the available SteeringWheels.

2 Right-click on the wheel and click SteeringWheel Settings.

3 In the SteeringWheels Settings dialog box, under Display, click Show the Pinned Wheel at Startup.

   The wheel is displayed as pinned when it is displayed; when left unchecked, the wheel follows the position of the cursor.

4 Click OK.

Ribbon: View tab ➤ Navigate panel ➤ SteeringWheels flyout

Menu: View ➤ SteeringWheels

Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels
Command entry: NAVSWHEEL

To enable tooltips for wheels

1. Click View tab ➤ Navigate panel ➤ SteeringWheels flyout ➤ click one of the available SteeringWheels.
2. Right-click on the wheel and click SteeringWheel Settings.
3. In the SteeringWheels Settings dialog box, under Display, click Show Tooltips.

   Tooltips are displayed for each wedge and buttons on a wheel when the cursor hovers over the wheel.
4. Click OK.

Menu: View ➤ SteeringWheels
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels
Command entry: NAVSWHEEL

To enable messages for wheels

1. Click View tab ➤ Navigate panel ➤ SteeringWheels flyout ➤ click one of the available SteeringWheels.
2. Right-click on the wheel and click SteeringWheel Settings.
3. In the SteeringWheels Settings dialog box, under Display, click Show Tool Messages.

   Messages are displayed when you use the navigation tools.
4. Click OK.

Menu: View ➤ SteeringWheels
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels
Command entry: NAVSWHEEL

Quick Reference

Commands

NAVSWHEEL

Displays a wheel that contains a collection of view navigation tools.

OPTIONS

Customizes the program settings.

System Variables

NAVSWHEELMODE

Specifies the current mode of the SteeringWheel.

NAVSWHEELOPACITYBIG

Controls the opacity of the big SteeringWheels.

NAVSWHEELOPACITYMINI

Controls the opacity of the mini SteeringWheels.

NAVSWHEELSIZEBIG

Specifies the size of the big SteeringWheels.

NAVSWHEELSIZEMINI

Specifies the size of the mini SteeringWheels.

Wheel Menu

From the Wheel menu, you can switch between different wheels and change the view of the model.

Use the Wheel menu to switch between the big and mini wheels that are available, go to the Home view, change the preferences of the current wheel, and control the behavior of the orbit, look, and walk 3D navigation tools. The menu items available on the Wheel menu are dependent on the current wheel and program.

The Wheel menu has the following options:

- **Mini View Object Wheel.** Displays the mini View Object wheel.
- **Mini Tour Building Wheel.** Displays the mini Tour Building wheel.

- **Mini Full Navigation Wheel.** Displays the mini Full Navigation wheel.
- **Full Navigation Wheel.** Displays the big Full Navigation wheel.
- **Basic Wheels.** Displays the big View Object or Tour Building wheel.
- **Go Home.** Goes to the Home view saved with the model.
- **Fit to Window.** Resizes and centers the current view to display all objects.
- **Restore Original Center.** Restores the center point of the view to the extents of the model.
- **Level Camera.** Rotates the current view so it is relative to the *XY* ground plane.
- **Increase Walk Speed.** Increases the walk speed used for the Walk tool by two times.
- **Decrease Walk Speed.** Decreases the walk speed used for the Walk tool by one half.
- **Help.** Launches the online Help system and displays the topic about the wheels.
- **SteeringWheel Settings.** Displays the dialog box where you can adjust the preferences for the wheels.
- **Close Wheel.** Closes the wheel.

To display the Wheel menu

- Click the down arrow in the lower-right corner of the wheel or right-click on the wheel.

Command entry: NAVSWHEEL
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels

Quick Reference

Commands

NAVSWHEEL

Displays a wheel that contains a collection of view navigation tools.

System Variables

NAVSWHEELMODE

Specifies the current mode of the SteeringWheel.

Navigation Wheels

You can choose from several different wheels. Some wheels are designed for 2D navigation while other wheels are better suited for 3D navigation.

2D Navigation Wheel

The 2D Navigation wheel is for basic navigation of 2D views.

With this wheel you can access basic 2D navigation tools; it is particularly useful when you do not have a pointing device with a scroll wheel. The wheel includes the Pan and Zoom tools.

The 2D Navigation wheel wedges have the following options:

- ***Pan.*** Repositions the current view by panning.
- ***Zoom.*** Adjusts the magnification of the current view.
- ***Rewind.*** Restores the most recent view orientation. You can move backward or forward by clicking and dragging left or right.

To switch to the 2D navigation wheel

- Click View tab ➤ Navigate panel ➤ SteeringWheels flyout ➤ 2D Wheel.

Command entry: NAVSWHEEL
Shortcut menu: SteeringWheels
Toolbar: Status bar ➤ SteeringWheels

Quick Reference

Commands

NAVSWHEEL

Displays a wheel that contains a collection of view navigation tools.

System Variables

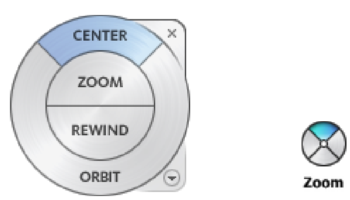
NAVSWHEELMODE

Specifies the current mode of the SteeringWheel.

View Object Wheels

The View Object wheels are used for 3D navigation. Use these wheels to view individual or groups of objects in a model.

With the View Object wheels (big and mini), you can view individual objects or features in a model. The big View Object wheel is optimized for new 3D users while the mini View Object wheel is optimized for experienced 3D users.



Big View Object Wheel

The big View Object wheel wedges have the following options:

- ***Center.*** Specifies a point on a model to adjust the center of the current view or change the target point used for some of the navigation tools.
- ***Zoom.*** Adjusts the magnification of the current view.
- ***Rewind.*** Restores the most recent view orientation. You can move backward or forward by clicking and dragging left or right.
- ***Orbit.*** Rotates the current view around a fixed pivot point.

Mini View Object Wheel

The mini View Object wheel wedges have the following options:

- ***Zoom (Top wedge).*** Adjusts the magnification of the current view.
- ***Rewind (Right wedge).*** Restores the most recent view. You can move backward or forward by clicking and dragging left or right.
- ***Pan (Bottom wedge).*** Repositions the current view by panning.
- ***Orbit (Left wedge).*** Rotates the current view around a fixed pivot point.

NOTE When the mini wheel is displayed, you can press and hold the middle mouse button to pan, scroll the wheel button to zoom in and out, and hold the SHIFT key while pressing and holding the middle mouse button to orbit the model.

To switch to the big View Objects wheel

Do one of the following:

- Click View tab ➤ Navigate panel ➤ SteeringWheels flyout ➤ Basic View Object Wheel.
- Right-click on the wheel, and click Basic Wheels ➤ View Object Wheel.

Command entry: NAVSWHEEL
Shortcut menu: SteeringWheels
Toolbar: Status bar ➤ SteeringWheels

To switch to the mini View Objects wheel

Do one of the following:

- Click View tab ➤ Navigate panel ➤ SteeringWheels flyout ➤ Mini View Object Wheel.
- Right-click on the wheel, and click Mini View Object Wheel.

Command entry: NAVSWHEEL
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels

Quick Reference

Commands

NAVSWHEEL

Displays a wheel that contains a collection of view navigation tools.

System Variables

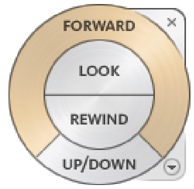
NAVSWHEELMODE

Specifies the current mode of the SteeringWheel.

Tour Building Wheels

The Tour Building wheels are for 3D navigation. Use these wheels to navigate within the interior of a model.

With the Tour Building wheels (big and mini), you can move through a model, such as a building, an assembly line, ship, or oil rig. You can also walk through and navigate around a model. The big Tour Building wheel is optimized for new 3D users while the mini Tour Building wheel is optimized for experienced 3D users.



Big Tour Building Wheel

The big Tour Building wheel wedges have the following options:

- ***Forward.*** Adjusts the distance between the current point of view and the defined pivot point of the model. Clicking once moves forward half the distance as far as the object you clicked.
- ***Look.*** Swivels the current view.
- ***Rewind.*** Restores the most recent view. You can move backward or forward by clicking and dragging left or right.
- ***Up/DownTool.*** Slides the current view of a model along the *Z* axis of the model.

Mini Tour Building Wheel

The mini Tour Building wheel wedges have the following options:

- ***Walk (Top wedge).*** Simulates walking through a model.
- ***Rewind (Right wedge).*** Restores the most recent view. You can move backward or forward by clicking and dragging left or right.
- ***Up/Down (Bottom wedge).*** Slides the current view of a model along the *Z* axis of the model.
- ***Look (Left wedge).*** Swivels the current view.

NOTE When the mini wheel is displayed, you can press and hold the middle mouse button to pan, scroll the wheel button to zoom in and out, and hold the SHIFT key while pressing and holding the middle mouse button to orbit the model.

To switch to the big Tour Building wheel

Do one of the following:

- Click View tab ➤ Navigate panel ➤ SteeringWheels flyout ➤ Basic Tour Building Wheel.
- Right-click on the wheel, and click Basic Wheels ➤ Tour Building Wheel.

Command entry: NAVSWHEEL

To switch to the mini Tour Building wheel

Do one of the following:

- Click View tab ➤ Navigate panel ➤ SteeringWheels flyout ➤ Mini Tour Building Wheel.
- Right-click on the wheel, and click Mini Tour Building Wheel.

Command entry: NAVSWHEEL
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels

Quick Reference

Commands

NAVSWHEEL

Displays a wheel that contains a collection of view navigation tools.

System Variables

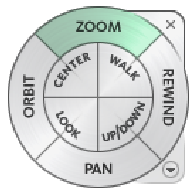
NAVSWHEELMODE

Specifies the current mode of the SteeringWheel.

Full Navigation Wheels

The Full Navigation wheels combine the 2D and 3D navigation tools found on the 2D Navigation, View Object, and Tour Building wheels into a single wheel.

The Full Navigation wheels (big and mini) combine the 3D navigation tools found on the View Object and Tour Building wheels. You can view individual objects, and walk through and around a model. The big and mini Full Navigation wheels are optimized for experienced 3D users.



NOTE When one of the Full Navigation wheels is displayed, you can press and hold the middle mouse button to pan, scroll the wheel button to zoom in and out, and hold the SHIFT key while pressing and holding the middle mouse button to orbit the model.

Big Full Navigation Wheel

The big Full Navigation wheel wedges have the following options:

- ***Zoom.*** Adjusts the magnification of the current view.
- ***Rewind.*** Restores the most recent view. You can move backward or forward by clicking and dragging left or right.
- ***Pan.*** Repositions the current view by panning.
- ***Orbit.*** Rotates the current view around a fixed pivot point.
- ***Center.*** Specifies a point on a model to adjust the center of the current view or change the target point used for some of the navigation tools.
- ***Walk.*** Simulates walking through a model.
- ***Look.*** Swivels the current view.
- ***Up/Down.*** Slides the current view of a model along the Z axis of the model.

Mini Full Navigation Wheel

The mini Full Navigation wheel wedges have the following options:

- ***Zoom (Top wedge).*** Adjusts the magnification of the current view.
- ***Walk (Upper right wedge).*** Simulates walking through a model.

- ***Rewind (Right wedge).*** Restores the most recent view. You can move backward or forward by clicking and dragging left or right.
- ***Up/Down (Lower right wedge).*** Slides the current view of a model along the Z axis of the model.
- ***Pan (Bottom wedge).*** Repositions the current view by panning.
- ***Look (Lower left wedge).*** Swivels the current view.
- ***Orbit (Left wedge).*** Rotates the current view around a fixed pivot point.
- ***Center (Upper left wedge).*** Specifies a point on a model to adjust the center of the current view or change the target point used for some of the navigation tools.

To switch to the big Full Navigation wheel

Do one of the following:

- Click View tab ➤ Navigate panel ➤ SteeringWheels flyout ➤ Full Navigation Wheel.
- Right-click on the wheel, and click Full Navigation Wheel.

Command entry: NAVSWHEEL
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels

To switch to the mini Full Navigation wheel

Do one of the following:

- Click View tab ➤ Navigate panel ➤ SteeringWheels flyout ➤ Mini Full Navigation Wheel.
- Right-click on the wheel, and click Mini Full Navigation Wheel.

Command entry: NAVSWHEEL
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels

Quick Reference

Commands

NAVSWHEEL

Displays a wheel that contains a collection of view navigation tools.

System Variables

NAVSWHEELMODE

Specifies the current mode of the SteeringWheel.

Navigation Tools

The navigation tools reorient the current view of a model.

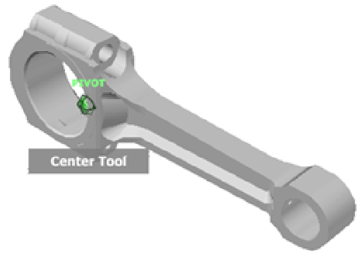
The display of a model can be adjusted by increasing or decreasing the magnification at which objects are displayed, rotating the model among other ways of changing the orientation of the model using the tools on SteeringWheels. You can create a view that defines an area of a model as the Home view and use preset views to restore known viewpoints of a model with the Autodesk® ViewCube® navigation tool.

Center Tool

The Center tool specifies a point on a model as the center of the current view. It also changes the target point used for some of the navigation tools.

With the Center tool, you can define the center of the current view of a model. To define the center, you drag the cursor over your model. A sphere is displayed in addition to the cursor. The sphere indicates that the point below the cursor in the model will be the center of the current view when you release the mouse button, the model is centered on the sphere.

NOTE If a center point on a model cannot be identified, then a prohibited icon (a circle with a diagonal line) is displayed instead of the sphere.



The point defined by the Center tool provides a focal point for the Zoom tool and a pivot point for the Orbit tool.

NOTE If you want to zoom from the Full Navigation wheels from your defined center point, hold down *CTRL* before zooming.

To specify a point on a model as the center of a view

1. Display one of the Full Navigation wheels or the big View Object wheel.
2. Click and hold down the Center wedge.
3. Drag the cursor to the desired location of the model.
4. Release the button on your pointing device when the sphere is displayed.

   The model is panned until the sphere is centered.

Command entry: NAVSWHEEL
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels

To specify the target point for the Zoom and Orbit tools

1. Display one of the Full Navigation wheels or the big View Object wheel.
2. Click and hold down the Center wedge.
3. Drag the cursor over the desired location of the model.
4. Release the button on your pointing device when the sphere is displayed.

   The model is panned until the sphere is centered.
5. Use the Zoom or Orbit tool to reorient the view of the model.

   If you are using one of the Full Navigation wheels, hold down the *CTRL* key before using the Zoom tool.

Command entry: NAVSWHEEL

Shortcut menu: Right-click over the drawing window and click SteeringWheels

Toolbar: Status bar ➤ SteeringWheels

Quick Reference

Commands

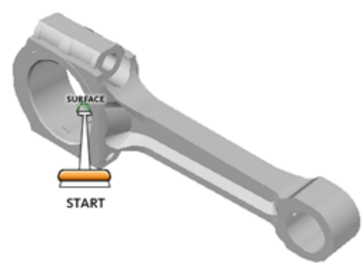
NAVSWHEEL

Displays a wheel that contains a collection of view navigation tools.

Forward Tool

The Forward tool adjusts the distance between the current point of view and the defined pivot point of the model.

You use the Forward tool to change the magnification of the model by increasing or decreasing the distance between the current point of view and the pivot point. The distance that you can move forward or backward is limited by the position of the pivot point.



To adjust the distance between the current point of view and the pivot point you use the Drag Distance indicator. The Drag Distance indicator has two marks on it that show the start and destination distances from the current point of view. The current traveled distance is shown by the orange position indicator. Slide the indicator forward or backwards to decrease or increase the distance towards the pivot point.

To reorient a view by moving towards or away from the model

1 Display the big Tour Building wheel.

2 Click and hold down the Forward wedge.

The Drag Distance indicator is displayed.

NOTE If you click the Forward wedge once, the model moves forward 50% of the distance between the current location and the pivot point.

---

3 Drag the cursor up or down to change the distance from which you view the model.

4 Release the button on your pointing device to return to the wheel.

Command entry: NAVSWHEEL
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels

Quick Reference

Commands

NAVSWHEEL

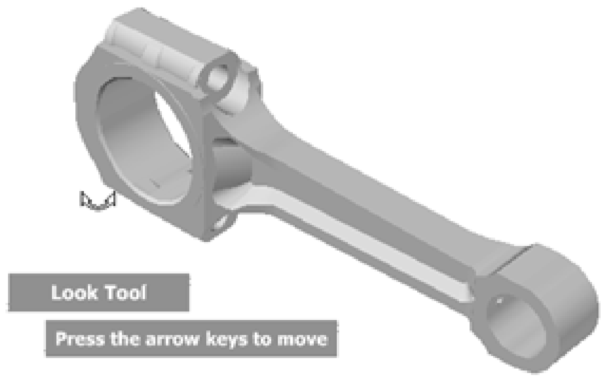
Displays a wheel that contains a collection of view navigation tools.

Look Tool

The Look tool rotates the view horizontally and vertically around a fixed point.

With the Look tool, you can rotate the current view vertically and horizontally. When rotating the view, your line of sight rotates about the current eye position, like turning your head. The Look tool can be compared to you standing in a fixed location, and looking up or down while turning your head left or right.

When using the Look tool, you adjust the view of the model by dragging the cursor. As you drag, the cursor changes to the Look cursor and the model rotates around the location of the current view.



Walking through a Model

When using the Look tool from the big Full Navigation wheel, you can walk through a model by using the arrow keys on the keyboard. Use the properties dialog box for the SteeringWheels to adjust the walk speed.

Invert Vertical Axis

When you drag the cursor upward, the target point of the view raises; dragging the cursor downward lowers the target point of the view. Use the properties dialog box for the SteeringWheels to invert the vertical axis for the Look tool.

To look around a view with the Look tool

1 Display one of the Full Navigation wheels or the mini Tour Building wheel.

2 Click and hold down the Look wedge.

   The cursor changes to the Look cursor.

3 Drag the pointing device to change the direction in which you are looking.

4 Release the button on your pointing device to return to the wheel.

Command entry: NAVSWHEEL
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels

To look around and walk through a model with the Look tool

1. Display the big Full Navigation wheel.
2. Click and hold down the Look wedge.

   The cursor changes to the Look cursor.
3. Drag to change the direction in which you are looking.
4. While holding down the button on your pointing device, press the arrow keys to walk in the model.
5. Release the button on your pointing device to return to the wheel.
6. Click Close to exit the wheel.

Command entry: NAVSWHEEL
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels

To invert the vertical axis for the Look tool

1. Click View tab ➤ Navigate panel ➤ SteeringWheels flyout ➤ click one of the available SteeringWheels 
2. Right-click on the wheel and click SteeringWheel Settings.
3. In the SteeringWheels Settings dialog box, click Invert Vertical Axis for Look Tool.

   Dragging the cursor downward lowers the target point of the view; dragging the cursor upward raises the target point of the view.
4. Click OK.

Command entry: NAVSWHEEL
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels

Quick Reference

Commands

NAVSWHEEL

Displays a wheel that contains a collection of view navigation tools.

Orbit Tool

The Orbit tool rotates the current view around a model based on a fixed pivot point.

You use the Orbit tool to change the orientation of a model. The cursor changes to the Orbit cursor. As you drag the cursor, the model rotates around a pivot point while the view remains fixed.

Specify the Pivot Point

The pivot point is the base point used when rotating the model with the Orbit tool. You can specify the pivot point in the following ways:

- **Default pivot point**. When you first open a model, the target point of the current view is used as the pivot point for orbiting the model.
- **Select objects**. You can select objects before the Orbit tool is used to calculate the pivot point. The pivot point is calculated based on the center of the extents of the selected objects.
- **Center tool**. You can specify a point on the model to use as the pivot point for orbiting with the *Center tool*.
- **CTRL+Click and drag**. Press and hold down the *CTRL* key before clicking the Orbit wedge or while the Orbit tool is active; then drag to the point on the model you want to use as the pivot point. This option is only available when using the big and mini Full Navigation wheels or the mini View Object wheel.

NOTE While the Orbit tool is active, you can be press and hold the *CTRL* key at anytime to move the pivot point used by the Orbit tool.

Maintain Up Direction

You can control how the model orbits around the pivot point by choosing to maintain the up direction of the model. When the up direction is maintained, orbiting is constrained along the *XY* axis and in the *Z* direction. If you drag horizontally, the camera moves parallel to the *XY* plane. If you drag vertically, the camera moves along the *Z* axis.

If the up direction is not maintained, you can roll the model using the roll ring which is centered around the pivot point. Use the properties dialog box for the SteeringWheels to control whether the up direction is maintained or not for the Orbit tool.

To orbit a model with the Orbit tool

1. Display one of the View Object or Full Navigation wheels.
2. Click and hold down the Orbit wedge.
   The cursor changes to the Orbit cursor.
3. Drag to rotate the model.

   NOTE Use the Center tool to re-center the model in the current view, if you are using one of the Full Navigation or View Object wheels.

4. Release the button on your pointing device to return to the wheel.

Command entry: NAVSWHEEL
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels

To orbit around an object with the Orbit tool

1. Press ESC to make sure no commands are active and to clear any previously selected objects.
2. Select the objects in the model for which you want to define the pivot point.
3. Display one of the View Object or Full Navigation wheels.

4 Click and hold down the Orbit wedge.

   The cursor changes to the Orbit cursor.

5 Drag to rotate the model.

6 Release the button on your pointing device to return to the wheel.

Command entry: NAVSWHEEL
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels

To turn on selection sensitivity for the Orbit tool

1 Click View tab ➤ Navigate panel ➤ SteeringWheels flyout ➤ click one of the View Object or Full Navigation wheels.

2 Right-click on the wheel and click SteeringWheel Settings.

3 In the SteeringWheels Settings dialog box, click Use Selection Sensitivity for Orbit Tool.

   The extents of any objects that are selected before the wheel is displayed are used to define the pivot point for the Orbit tool. If no objects are selected, the pivot point used by the Orbit tool is the one defined by the Center tool.

4 Click OK.

Command entry: NAVSWHEEL
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels

To maintain the up direction for the Orbit tool

1 Click View tab ➤ Navigate panel ➤ SteeringWheels flyout ➤ click one of the View Object or Full Navigation wheels.

2 Right-click on the wheel and click SteeringWheel Settings.

3 In the SteeringWheels Settings dialog box, click Maintain Up Direction for Orbit Tool.

Orbiting the model is constrained along the *XY* plane and *Z* directions.

4 Click OK.

Command entry: NAVSWHEEL
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels

Quick Reference

Commands

NAVSWHEEL

Displays a wheel that contains a collection of view navigation tools.

Pan Tool

The Pan tool repositions the current view of the model by panning.

When the pan tool is active, the Pan cursor (a four-sided arrow) is displayed. Dragging the pointing device moves the model in the same direction. For example, dragging upward moves the model up while dragging downward moves the model down.

TIP If the cursor reaches the edge of the screen, you can continue panning by dragging further to force it to wrap around the screen.

To pan the view with the Pan tool

1 Display the 2D Navigation wheel, one of the Full Navigation wheels, or the mini View Object wheel.

2 Click and hold the Pan wedge.

The cursor changes to the Pan cursor.

3 Drag to reposition the model.

4 Release the button on your pointing device to return to the wheel.

Ribbon: View tab ➤ Navigate panel ➤ SteeringWheels flyout
Menu: View ➤ SteeringWheels
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels
Command entry: NAVSWHEEL

To start the Pan tool with the middle button

1 Display the 2D Navigation wheel, the Full Navigation wheel, or one of the mini wheels.

2 Press and hold down the scroll wheel or middle button.

The cursor changes to the Pan cursor.

3 Drag to reposition the model.

4 Release the wheel or button on your pointing device to return to the wheel.

Ribbon: View tab ➤ Navigate panel ➤ SteeringWheels flyout
Menu: View ➤ SteeringWheels
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels
Command entry: NAVSWHEEL

Quick Reference

Commands

NAVSWHEEL

Displays a wheel that contains a collection of view navigation tools.

Rewind Tool

The Rewind tool restores the most recent view. You can also move backward or forward through previous views.

As you use the navigation tools to reorient the view of a model, the previous view is saved to the navigation history. The navigation history holds a representation of the previous views of the model along with a thumbnail. A separate navigation history is maintained for each window; it is not maintained after the window is closed. Rewind navigation history is view-specific.

With the Rewind tool, you can retrieve previous views from the navigation history. From the navigation history, you can restore a previous view or scroll through all of the saved views.

When you hold down the button on the pointing device over the Rewind tool on the wheel, the Rewind History panel is displayed. You can scroll through the navigation history. To restore one of the previous views in the navigation history, drag the bracket to the left in the Rewind History panel.

When a view change occurs, the previous view is recorded to the navigation history. If the view change is made with a wheel, a thumbnail is automatically generated and added to the Rewind UI. For view changes that are made not using a wheel, a thumbnail is only generated when the system variable *CAPTURETHUMBNAILS* is set to a value of 2. You can control when thumbnails are generated for view changes in the SteeringWheels Settings dialog box.

To restore the previous view

1 Display a wheel.

2 Click the Rewind wedge.

Ribbon: View tab ➤ Navigate panel ➤ SteeringWheels flyout 
Menu: View ➤ SteeringWheels
Shortcut menu: Right-click over the drawing window and click SteeringWheels

Toolbar: Status bar ➤ SteeringWheels
Command entry: NAVSWHEEL

To restore a previous view with the Rewind History panel

1. Display a wheel.
2. Click and hold the Rewind wedge.

   The Rewind History panel is displayed.
3. While holding down the button on your pointing device, drag to the left or to the right to restore a previous view.

   Dragging to the left restores an older previous view. Dragging to the right restores a view that is newer than the one you are currently viewing. You must have previously used the Rewind tool to see views available on the right. The current position in the navigation history is indicated by the orange box that is dragged along the Rewind History panel.

Ribbon: View tab ➤ Navigate panel ➤ SteeringWheels flyout
Menu: View ➤ SteeringWheels
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels
Command entry: NAVSWHEEL

To specify the display of rewind thumbnails

1. Display a wheel.
2. Right-click on the wheel and click SteeringWheel Settings.
3. In the SteeringWheels Settings dialog box, select one of the following:
   - **Never** - Only display thumbnails for view changes made with the SteeringWheels.
   - **On Demand When the Bracket Is Moved Over an Empty Frame** - Display thumbnails on demand for a previous view change when using the Rewind UI.
   - **Automatically When a View Change Occurs** - Display thumbnails for all view changes saved to the navigation history.
4. Click OK.

Ribbon: View tab ➤ Navigate panel ➤ SteeringWheels flyout
Menu: View ➤ SteeringWheels
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels
Command entry: NAVSWHEEL

Quick Reference

Commands

NAVSWHEEL

Displays a wheel that contains a collection of view navigation tools.

System Variables

CAPTURETHUMBNAILS

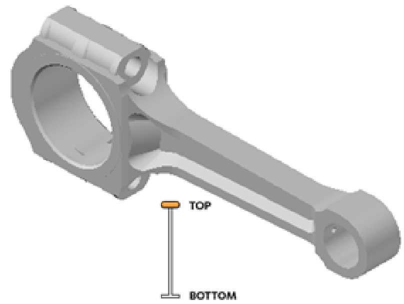
Specifies if and when thumbnails are captured for the Rewind tool.

Up/Down Tool

The Up/Down tool slides the current view of a model along the *Z* axis of the model.

Unlike the Pan tool, you use the UP/Down tool to adjust the height of the current viewpoint along the model's Z axis. To adjust the vertical elevation of the current view, you drag up or down. As you drag, the current elevation and the allowed range of motion is displayed on a graphical element called the Vertical Distance indicator.

The Vertical Distance indicator has two marks that show the highest (Top) and lowest (Bottom) elevation the view can have. While changing the elevation with the Vertical Distance indicator, the current elevation is shown by the bright orange indicator, while the previous elevation is shown by the dim orange indicator.



To change the elevation of a view

1. Display one of the Full Navigation wheels or the Tour Building wheels.
2. Click and hold down the Up/Down wedge.
   The Vertical Distance indicator is displayed.
3. Drag up or down to change the elevation of the view.
4. Release the button on your pointing device to return to the wheel.

Ribbon: View tab ➤ Navigate panel ➤ SteeringWheels flyout 
Menu: View ➤ SteeringWheels
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels
Command entry: NAVSWHEEL

Quick Reference

Commands

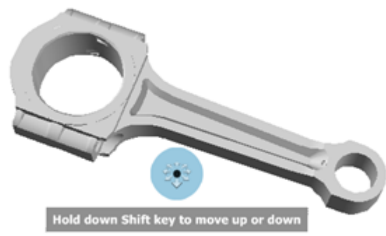
NAVSWHEEL

Displays a wheel that contains a collection of view navigation tools.

Walk Tool

The Walk tool simulates walking through a model.

With the Walk tool, you can navigate through a model as if you were walking through it. Once you start the Walk tool, the Center Circle icon is displayed near the center of the view and the cursor changes to display a series of arrows. To walk through the model, you drag in the direction in which you want to move in.



Constrain the Walk Angle

When walking through a model, you can constrain the movement angle to the ground plane. If the Constrain Walk Angle to Ground Plane option is enabled, you can freely walk around while maintaining a constant camera viewpoint elevation; if the walk angle is not constrained, you will “fly” in the direction you are looking. Use the properties dialog box for the SteeringWheels to constrain the movement angle to the ground plane for the Walk tool.

Movement Speed

As you walk or “fly” through a model, you can control the movement speed. Movement speed is controlled by the distance in which the Cursor is moved from the Center Circle icon and the current movement speed setting. You can adjust the movement speed setting permanently and temporarily as you use the Walk tool. To temporarily increase movement speed, press and hold the + (plus) key while using the Walk tool.

Change the Elevation

As you use the Walk tool, you can adjust the camera elevation by holding down the *SHIFT* key. This temporarily activates the Up/Down tool. With the Up/Down tool active, drag up or down to adjust the elevation of the camera. You can also use the *UP ARROW* and *DOWN ARROW* keys as you walk to adjust the height of the view.

To use the Walk tool to move through the model

1 Display one of the Full Navigation wheels or the mini Tour Building wheel.

2 Click and hold down the Walk wedge.

   The cursor changes to the Walk cursor and the Center Circle icon is displayed.

3 Drag in the direction you want to walk.

   NOTE While walking, press and hold down the + (plus) key to temporarily increase your movement speed.
   ---

4 Release the button on your pointing device to return to the wheel.

Ribbon: View tab ➤ Navigate panel ➤ SteeringWheels flyout 
Menu: View ➤ SteeringWheels
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels
Command entry: NAVSWHEEL

To change the movement speed for the Walk tool

1 Display a wheel.

2 Right-click on the wheel and click SteeringWheel Settings.

3 In the SteeringWheels Settings dialog box, under Walk Tool, drag the Walk Speed slider to the left to decrease the walking speed or to the right to increase the walking speed.

4 Click OK.

Ribbon: View tab ➤ Navigate panel ➤ SteeringWheels flyout 
Menu: View ➤ SteeringWheels
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels
Command entry: NAVSWHEEL

To constrain the walk angle to the ground plane

1 Display a wheel.

2 Right-click on the wheel and click SteeringWheel Settings.

3 In the SteeringWheels Settings dialog box, under Walk Tool, click Constrain Walk Angle to Ground Plane.

   Movement when walking is done parallel to the ground plane of the model.

4 Click OK.

Ribbon: View tab ➤ Navigate panel ➤ SteeringWheels flyout
Menu: View ➤ SteeringWheels
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels
Command entry: NAVSWHEEL

To adjust the height of the current view from the Walk tool

1 Display one of the Full Navigation wheels or the mini Tour Building wheel.

2 Click and hold down the Walk wedge.

   The cursor changes to the Walk cursor and the Center Circle icon is displayed.

3 Do one of the following:

   - Press and hold down the *SHIFT* key to enable the Up/Down tool; drag up or down.
   - Press and hold down the *UP ARROW* or *DOWN ARROW* key.

4 Release the button on your pointing device to return to the wheel.

Ribbon: View tab ➤ Navigate panel ➤ SteeringWheels flyout
Menu: View ➤ SteeringWheels
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels
Command entry: NAVSWHEEL

Quick Reference

Commands

NAVSWHEEL

Displays a wheel that contains a collection of view navigation tools.

Zoom Tool

The Zoom tool adjusts the magnification of the current view of a model.

You use the Zoom tool to change the zoom magnification of a model. The following mouse click and key combinations are available to control how the Zoom tool behaves:

- **Click.** If you click the Zoom tool on a wheel, the current view is zoomed in by a factor of 25 percent. If you are using the Full Navigation wheel, incremental zoom must be enabled in the properties dialog box for the SteeringWheels.
- **SHIFT+click.** If you hold down the *SHIFT* key before you click the Zoom tool on a wheel, the current view is zoomed out by a factor of 25 percent. Zooming is performed from the current location of the cursor, and not the current pivot point.

  NOTE When you start the Zoom tool from the Full Navigation wheel, incremental zooming must be enabled in the properties dialog box for the SteeringWheels in order to use *CTRL*+click and *SHIFT*+click.

- **CTRL+click.** If you hold down the *CTRL* key before you click the Zoom tool on a wheel, the current view is zoomed in by a factor of 25 percent. Zooming is performed from the current pivot point, and not the location of the cursor.
- **Click and drag.** If you click the Zoom tool and hold down the button on your pointing device, you can adjust the magnification of the model by dragging up and down.
- **CTRL+click and drag.** When using the Full Navigation wheels or the mini View Object wheel, you can control the target point used by the Zoom tool. By holding down the *CTRL* key, the Zoom tool uses the location of the previous pivot point defined by the Zoom, Orbit, or Center tool.

NOTE When you start the Zoom tool from the Full Navigation wheel, incremental zooming must be enabled in the properties dialog box for the SteeringWheels in order to use *CTRL*+click and *SHIFT*+click.

NOTE When you use the Zoom tool from the Full Navigation wheel or the View Object wheel, the point in the view where you click to zoom becomes the Center point for future Orbit operations until you either use the Zoom tool again or use the Center tool. If you press *CTRL* before you click the Zoom wedge, the Center point does not change.

Zoom Constraints

When changing the magnification of a model with the Zoom tool, you cannot zoom in any further than the focus point or out past the extents of the model. The direction you can zoom in and out is controlled by the center point set by the Center tool.

When changing the magnification of a model with the Zoom tool, you cannot zoom in any further than the focus point or out past the extents of the model.

To zoom a view with a single click

1. Display a wheel.
2. Right-click on the wheel and click SteeringWheel Settings.
3. In the SteeringWheels Settings dialog box, under Zoom Tool, select Enable Single Click Incremental Zoom.
4. Click OK.
5. Display one of the Full Navigation Wheels or the mini View Object Wheel.
6. Click the Zoom wedge.

   The magnification of the model is increased and you are zoomed in closer to the model. If you hold down the Shift key while clicking the Zoom wedge, the model is zoomed out or you can hold down the Ctrl key to zoom in.

7 Click Close to exit the wheel.

Ribbon: View tab ➤ Navigate panel ➤ SteeringWheels flyout
Menu: View ➤ SteeringWheels
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels
Command entry: NAVSWHEEL

To zoom a view in and out by dragging

1 Display the 2D Navigation wheel, one of the Full Navigation wheels, or the mini View Object wheel.

2 Click and hold down the Zoom wedge.

The cursor changes to the Zoom cursor.

3 Drag vertically to zoom in or out.

4 Release the button on your pointing device to return to the wheel.

Ribbon: View tab ➤ Navigate panel ➤ SteeringWheels flyout
Menu: View ➤ SteeringWheels
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels
Command entry: NAVSWHEEL

To zoom in and out by scrolling the mouse wheel when a SteeringWheel is displayed

1 Display one of the wheels other than the big Tour Building wheel.

2 Scroll the wheel forward or backward to zoom in or out.

3 Release the button on your pointing device to return to the wheel.

Ribbon: View tab ➤ Navigate panel ➤ SteeringWheels flyout
Menu: View ➤ SteeringWheels
Shortcut menu: Right-click over the drawing window and click SteeringWheels
Toolbar: Status bar ➤ SteeringWheels
Command entry: NAVSWHEEL

Quick Reference

Commands

NAVSWHEEL

Displays a wheel that contains a collection of view navigation tools.

Define and Change Views with ShowMotion

The Autodesk® ShowMotion® navigation tool provides an onscreen display that can be used to create and play back cinematic camera animations. These animations can be used for presentation purposes or to navigate through a design.

Overview of ShowMotion

Using ShowMotion, you can add movement and transitions to a saved view. These saved views are called shots. The types of shots that you can create are:

- **Still.** Utilizes a single fixed camera position.
- **Cinematic.** Utilizes a single camera with cinematic camera movements.
- **Recorded Walk.** Records an animation by navigating around and through a model.

Shot Sequences

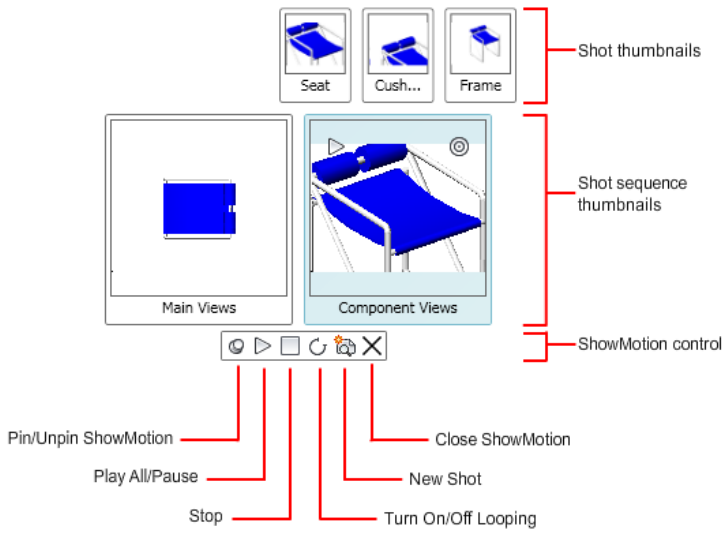
Shot sequences are used to organize related shots together. These sequences are also called view categories. By using shot sequences, you can do the following:

- Quickly locate a shot

- Play back more than one shot at a time
- Control the order in which shots are played back

Using ShowMotion

ShowMotion is made up of three main parts: shot thumbnails, shot sequence thumbnails, and the ShowMotion control. With the ShowMotion control, which is along the bottom, you can play the animations assigned to a shot, pin and unpin ShowMotion, and close ShowMotion. Use the shot and shot sequence thumbnails to navigate the shots in the current model.



The ShowMotion control has the following options:

- **Pin/Unpin ShowMotion.** Pins ShowMotion so the ShowMotion control and all thumbnails remain displayed even if focus is shifted away from ShowMotion. When ShowMotion is not pinned, the ShowMotion control and all thumbnails disappear if the focus is switched away from ShowMotion.
- **Play All.** Starts the playback of shots in all shot sequences. Shots are played left to right, starting with the leftmost shot sequence.
- **Stop.** Stops the playback of the current shot.
- **Turn On/Off Looping.** Enables or disables playback looping for the animation assigned to the shot or shot sequence when played back.

- **New Shot.** Displays the New View/Shot Properties dialog box where you can create a new shot.
- **Close ShowMotion.** Closes the ShowMotion control and all thumbnails.

To launch ShowMotion

Use one of the following methods to launch ShowMotion

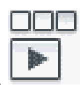
- Click View ➤ ShowMotion.
- On the status bar, click ShowMotion.

Command entry: NAVSMOTION

To pin or unpin ShowMotion

- On the ShowMotion control, click Pin ShowMotion or Unpin ShowMotion.

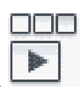
Menu: View ➤ ShowMotion

Toolbar: Status bar ➤ ShowMotion 
Command entry: NAVSMOTION

To close ShowMotion

- On the ShowMotion control, click Close ShowMotion or click in the document window when ShowMotion is not pinned.

Menu: View ➤ ShowMotion

Toolbar: Status bar ➤ ShowMotion 
Command entry: NAVSMOTION

Quick Reference

Commands

NAVSMOTION

Displays the ShowMotion interface.

Create and Modify Shots and Shot Sequences

You create and modify shots from ShowMotion. As you create or modify a shot, a thumbnail is generated and added to a shot sequence.

Create and Modify a Shot

When a shot is created, it must be given a name and view type. The view type assigned to the shot determines which transition and motion options you can change. After a shot is created, a thumbnail is automatically generated and placed under the shot sequence that was assigned to the shot. The name of the shot is located below the thumbnail. If you need to change a shot, you can right-click the shot that you want to modify.

The shortcut menu of a shot has the following options:

- **Properties.** Displays the dialog box to modify the shot's transition and motion settings, and assigned shot sequence, among other settings.
- **Rename.** Renames a shot.
- **Delete.** Removes a shot.
- **Move Left** and **Move Right.** Changes the position of a shot in the shot sequence by one position to the left or right.
- **Update the Thumbnail.** Updates the thumbnail for a single shot or for all shots saved with the model.

Create and Modify a Shot Sequence

When a shot is created, you can add the shot to the default shot sequence or specify the shot sequence to which the shot should be added. Each shot sequence is represented by a thumbnail in ShowMotion. The thumbnail is identical to the first shot in the shot sequence. The name of the shot sequence is below the thumbnail.

The shortcut menu of a shot sequence has the following options:

- **Rename.** Renames a shot sequence.
- **Delete.** Removes a shot sequence.
- **Move Left** and **Move Right.** Changes the position of a shot sequence in ShowMotion by one position to the left or right.
- **Update the Thumbnail.** Updates the thumbnails for all shots in a shot sequence or for all shots saved with the model.

To create a still shot

1 On the ShowMotion control, click New Shot.

2 In the New View/Shot Properties dialog box, in the View Name text box, enter a name.

3 In the View Category drop-down list, select a view category.

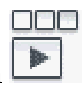
4 In the View Type drop-down list, select a shot type.

5 On the Shot Properties tab, under Transition, select a transition type from the Transition Type drop-down list.

6 Under Transition, enter a transition duration in the Transition Duration text box.

7 Click OK.

Menu: View ➤ ShowMotion

Toolbar: Status bar ➤ ShowMotion 
Command entry: NAVSMOTION

To create a recorded walk shot

1 On the ShowMotion control, click New Shot.

2 In the New View/Shot Properties dialog box, in the View name box, enter a name.

3 In the View Category drop-down list, select a view category.

4 In the View Type drop-down list, select Recorded Walk.

5 On the Shot Properties tab, under Transition, select a transition type from the Transition Type drop-down list.

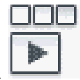
6 Under Transition, enter a transition duration in the Transition Duration text box.

7 Under Motion, click Start Recording.

8 Click on the 3D canvas and drag the mouse along the desired path of animation. Release the mouse to stop recording.

9 Click OK.

Menu: View ➤ ShowMotion

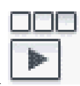
Toolbar: Status bar ➤ ShowMotion 

Command entry: NAVSMOTION

To create a cinematic shot

1. On the ShowMotion control, click New Shot.
2. In the New View/Shot Properties dialog box, in the View name box, enter a name.
3. In the View Category drop-down list, select a view category.
4. In the View Type drop-down list, select Cinematic.
5. On the Shot Properties tab, under Transition, select a transition type from the Transition Type drop-down list.
6. Under Transition, enter a transition duration in the Transition Duration text box.
7. Under Motion, set the camera position.
8. Under Motion, set the duration of the movement.
9. Under Motion, set the travel distance of the camera.
10. Click OK.

Menu: View ➤ ShowMotion

Toolbar: Status bar ➤ ShowMotion 

Command entry: NAVSMOTION

To modify a shot

1. On ShowMotion, hover the cursor over the shot sequence that contains the shot you want to modify.
2. Right-click the shot you want to modify, and click Properties.
3. In the View/Shot Properties dialog box, change the desired settings.
4. Click OK.

Menu: View ➤ ShowMotion

Toolbar: Status bar ➤ ShowMotion
Command entry: NAVSMOTION

To rename a shot or shot sequence

1. On ShowMotion, hover the cursor over the shot or shot sequence you want to rename.
2. Right-click, and click Rename.
3. In the in-place editor, replace the existing name with the new name and press ENTER.

Menu: View ➤ ShowMotion

Toolbar: Status bar ➤ ShowMotion
Command entry: NAVSMOTION

To resequence a shot or shot sequence

1. On ShowMotion, hover the cursor over the shot or shot sequence you want to resequence.
2. Right-click, and click Move Left or Move Right.

Menu: View ➤ ShowMotion

Toolbar: Status bar ➤ ShowMotion
Command entry: NAVSMOTION

To delete a shot

1. On ShowMotion, hover the cursor over the shot you want to delete.
2. Right-click, and click Delete.

Menu: View ➤ ShowMotion

Toolbar: Status bar ➤ ShowMotion
Command entry: NAVSMOTION

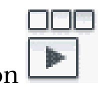
To delete a shot sequence

1 On ShowMotion, hover the cursor over the shot sequence you want to delete.

2 Right-click, and click Delete.

3 In the ShowMotion - Delete View Category dialog box, click Delete Category.

Menu: View ➤ ShowMotion

Toolbar: Status bar ➤ ShowMotion 

Command entry: NAVSMOTION

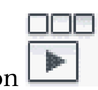
To update the thumbnail for a shot or shot sequence

1 On ShowMotion, hover the cursor over the shot or shot sequence for which you want to update the thumbnail.

2 Right-click, and click Update the Thumbnail For ➤ This View or This Category.

Click All to update the thumbnails for all shots stored in the model.

Menu: View ➤ ShowMotion

Toolbar: Status bar ➤ ShowMotion 

Command entry: NAVSMOTION

Quick Reference

Commands

NAVSMOTION

Displays the ShowMotion interface.

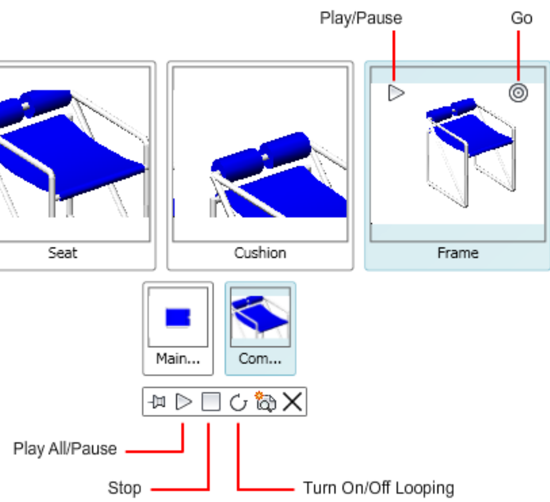
NEWSHOT

Creates a named view with motion that is played back when viewed with ShowMotion.

Play Back a Shot

You can view shots individually or together in a shot sequence.

Once a shot is created, you can play back the animation assigned to the shot by using the view controls located on each shot or shot sequence thumbnail or the ShowMotion control. The available view controls are:



- **Play.** Starts the playback of a shot or shot sequence. When the cursor is over the thumbnail, the Play button is displayed in the upper-left corner. The Play button plays back the individual shot or all the shots in the shot sequence.
- **Play All.** Starts the playback of all shots in all shot sequences. Shots are played left to right, starting with the leftmost shot sequence.
- **Pause.** Stops the playback of a shot or shot sequence.
- **Stop.** Stops the playback of the current shot.
- **Go.** Restores the view of the model assigned to a shot or the first shot in a shot sequence.

To play a shot

1 On ShowMotion, hover the cursor over the shot sequence that contains the shot you want to play.

2 Position the cursor over the thumbnail of the shot you want to view.

3 On the shot thumbnail, in the upper-left corner, click the Play button.

Menu: View ➤ ShowMotion

Toolbar: Status bar ➤ ShowMotion

Command entry: NAVSMOTION

To play all shots within a shot sequence

1 On ShowMotion, hover the cursor over the shot sequence that contains the shots you want to play.

2 On the thumbnail for the shot sequence, click the Play button located on the upper-left corner of the thumbnail.

   All the shots in the shot sequence are played.

Menu: View ➤ ShowMotion

Toolbar: Status bar ➤ ShowMotion

Command entry: NAVSMOTION

To play all shots within all shot sequences

- On the ShowMotion control, click Play All.
  All the shots saved in the model are played back in the order of each shot sequence.

Menu: View ➤ ShowMotion

Toolbar: Status bar ➤ ShowMotion

Command entry: NAVSMOTION

To pause or stop the playback of a shot

- On the ShowMotion control, click Pause to pause the playback of the current shot or Stop to stop playback.

Menu: View ➤ ShowMotion

Toolbar: Status bar ➤ ShowMotion

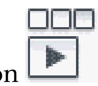
Command entry: NAVSMOTION

To loop the playback of a shot

- On the ShowMotion control, click Turn On/Off Looping.

NOTE If the loop setting is on, the shot loops until you click the Stop button or press the ESC key.

Menu: View ➤ ShowMotion

Toolbar: Status bar ➤ ShowMotion 

Command entry: NAVSMOTION

Quick Reference

Commands

NAVSMOTION

Displays the ShowMotion interface.

VIEWGO

Restores a named view.

VIEWPLAY

Plays the animation associated to a named view.

Define a 3D View with a Camera

Define a 3D view by placing a camera in model space and adjusting camera settings to suit your needs.

Overview of Cameras

You can place a camera in a drawing to define a 3D view.

You can turn a camera on or off in a drawing and use grips to edit a camera's location, target, or lens length. A camera is defined by a location *XYZ* coordinate, a target *XYZ* coordinate, and a field of view/lens length, which determines the magnification, or zoom factor. You can also define clipping planes, which establish front and back boundaries for the associated view.

- **Location**. Defines the point from which you are viewing a 3D model.
- **Target**. Defines the point you are viewing by specifying the coordinate at the center of the view.
- **Lens length**. Defines the magnification properties of a camera's lens. The greater the lens length, the narrower the field of view.
- **Front and back clipping planes**. Specifies the location of clipping planes. Clipping planes are boundaries that define, or clip, a view. In the camera's view, everything between the camera and the front clipping plane is hidden. Likewise, everything between the back clipping plane and the target is hidden.

By default, saved cameras are given names such as Camera1, Camera2, and so on. You can rename a camera to better describe its view. The View Manager lists existing cameras in a drawing as well as other named views.

Use the Camera Glyph Appearance dialog box to control the camera glyph's colors and size.

Quick Reference

Commands

CAMERA

Sets a camera and target location to create and save a 3D perspective view of objects.

OPTIONS

Customizes the program settings.

System Variables

CAMERADISPLAY

Turns the display of camera objects on or off.

CAMERAHEIGHT

Specifies the default height for new camera objects.

Create a Camera

Set a camera and target location to create and save 3D perspective views of objects.

You can create a new camera by defining its location and a target, and by further defining its name, height, lens length, and clipping planes. You can also use one of several pre-defined camera types that are available on the tool palette.

To create a camera

1 Click Render tab ➤ Camera panel ➤ Create Camera. 

2 Click in the drawing to specify the camera's location.

3 Click again in the drawing to specify a target location.

4 Do one of the following:
   - If you are finished setting up the camera, press Enter.
   - To further define camera properties, right-click and select from the list of options. Then press Enter to finish setting up the camera.

   CAMERA

   View 

To create a camera from the Tool palette

1 Click View tab ➤ Palettes panel ➤ Tool Palettes. 

2 In the Tool Palettes window, click the Camera tool palette tab to make it active.

3 Select a camera type. Drag the camera icon from the tool palette, and click in the drawing where you want to set its location.

4 Click again in the drawing when you want to place the target.

Command entry: TOOLPALETTES, CAMERA

To display a camera

- Click Render tab ➤ Camera panel ➤ Show Cameras.

Command entry: CAMERADISPLAY

Menu: Click View menu ➤ Display ➤ Cameras.

Quick Reference

Commands

CAMERA

Sets a camera and target location to create and save a 3D perspective view of objects.

OPTIONS

Customizes the program settings.

System Variables

CAMERADISPLAY

Turns the display of camera objects on or off.

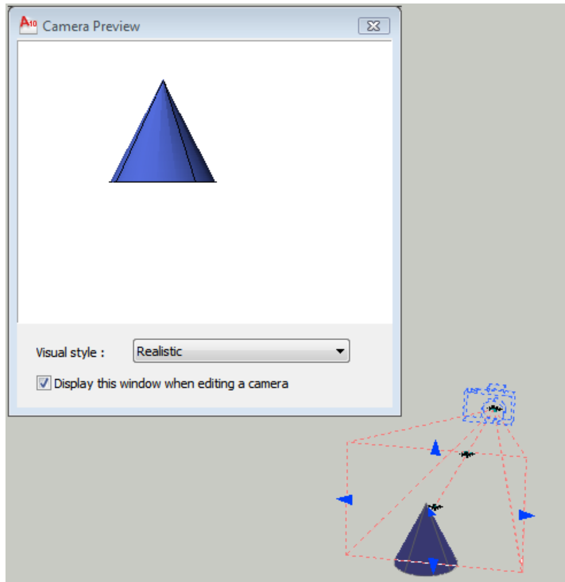
CAMERAHEIGHT

Specifies the default height for new camera objects.

Change Camera Properties

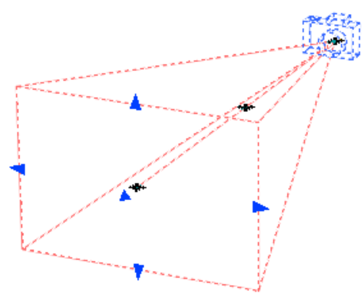
You can modify a camera’s lens length, change its front and back clipping planes, name a camera, and turn the display of all cameras on or off in a drawing.

When you select a camera, the Camera Preview dialog box opens to display the camera's view.

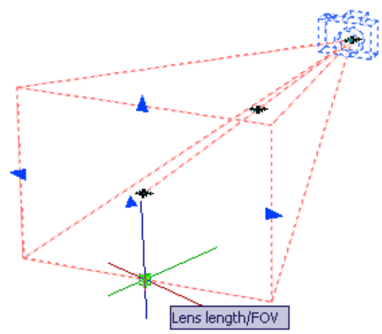


There are several ways to change camera settings:

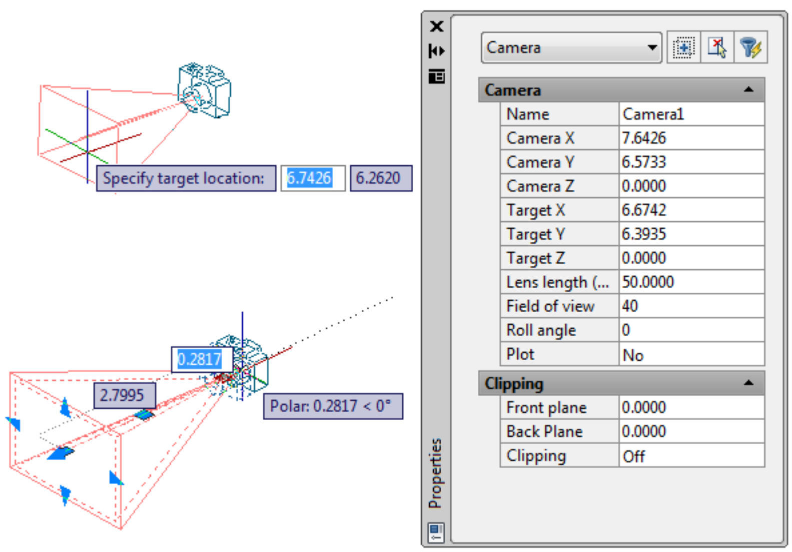
- Click and drag grips to resize or relocate a lens length or field of view.



- Use the Dynamic Input tooltip to enter *X*,*Y*, *Z* coordinate values.



- Modify camera properties in the Camera Properties palette.



To change a camera’s lens length

1 If cameras are not already displayed in the drawing, click Render tab ➤ Camera panel ➤ Show Cameras. 

2 Click the camera glyph.

3 Click a Lens Length/FOV grip tool.

4 Move the cursor and click where you want to position the lens.

5 Press Enter.

NOTE To change a lens length to a precise value, double-click a camera glyph to open the Properties palette. In the Cameras section, Lens Length option (mm), enter a numeric value.

Command entry: CAMERADISPLAY

To set a camera’s clipping planes

1 If cameras are not already displayed in the drawing, click Render tab ➤ Camera panel ➤ Show Cameras. 

2 Double-click the camera whose clipping planes you want to set.

3 In the Properties palette, Clipping section, Clipping option, select Front On, Back On, or Front and Back On.

4 In the Front Plane or Back Plane option, enter numeric values.

5 Press Enter.
CAMERADISPLAY

To rename a camera

1 If cameras are not already displayed in the drawing, click Render tab ➤ Camera panel ➤ Show Cameras. 

2 Double-click a camera glyph.

3 In the Properties palette, General section, Name option, enter a new name.

4 Press Enter.

Command entry: CAMERADISPLAY

To change a camera's location

1 If cameras are not already displayed in the drawing, click Render tab ➤ Camera panel ➤ Show Cameras. 

2 Click the camera glyph, drag it to the new location, and click to place the camera.

3 Press Enter.

Command entry: CAMERADISPLAY

To change a camera's target

1 If cameras are not already displayed in the drawing, click Render tab ➤ Camera panel ➤ Show Cameras. 

2 Click the camera glyph whose target you want to change.

3 Click the Target Distance grip tool (the blue grip in the center), drag it to the new location, and click to place the target.

4 Press Enter.

Command entry: CAMERADISPLAY

To specify whether the camera glyph plots

1 If cameras are not already displayed in the drawing, click Render tab ➤ Camera panel ➤ Show Cameras. 

2 Double-click a camera.

3 In the Properties palette, Camera section, Plot option, click Yes or No.

Command entry: CAMERADISPLAY

Quick Reference

Commands

CAMERA

Sets a camera and target location to create and save a 3D perspective view of objects.

System Variables

CAMERADISPLAY

Turns the display of camera objects on or off.

CAMERAHEIGHT

Specifies the default height for new camera objects.

Create Preview Animations

You can create 3D preview animations and adjust the settings before you create a motion path animation.

Preview animations are created with the controls on the Animation panel found on the ribbon on page 22 and the 3D navigation tools. Once a 3D navigation tool is active, the controls on the Animation panel are enabled to start recording an animation.

The following commands can be used to create an animation

- 3DCORBIT
- 3DDISTANCE
- 3DFLY
- 3DFORBIT
- 3DORBIT
- 3DPAN
- 3DSWIVEL
- 3DWALK

- 3DZOOM

To learn more about the 3D navigation tools available for creating an animation, see Use 3D Navigation Tools on page 323.

To create a preview animation

1. If the Animations panel is not displayed on the Render tab, right-click the Render tab and click Panels ➤ Animations.
2. Start any 3D navigation command such as 3DORBIT.
3. Click Render tab ➤ Animations panel ➤ Animation Record.
4. Navigate through the drawing. If you are in walk or fly mode, you can use the Position Locator window as a visual guide.
5. (Optional) Do any of the following:
   - Right-click the drawing and click Other Navigation Modes. Click another navigation mode.
   - Click Render tab ➤ Animations panel ➤ Animation Pause to set up another navigation mode or adjust animation settings. If you are in walk or fly mode, you can also adjust settings for the Position Locator window.
6. To review the animation, click Render tab ➤ Animations panel ➤ Animation Play. In the Animation Preview dialog box, view the animation, and do the following:
   - If you want to preview the animation in a different visual style, select another style from the drop-down list.
   - If you are satisfied with the animation playback, click the Pause button. Then click the Save button.
7. In the Save As dialog box, select a file location, file name, and file type (AVI, MPG, MOV, or WMV).
8. Click Save.

To record a preview animation

1. If the Animations panel is not displayed on the Render tab, right-click the Render tab and click Panels ➤ Animations.
2. Start any navigation command such as 3DORBIT.

3 Click Render tab ➤ Animations panel ➤ Animation Record.

4 Navigate through the model to record the movement.

5 When you are finished recording the animation, click Render tab ➤ Animations panel ➤ Animation Pause.

6 To review the animation, click Render tab ➤ Animations panel ➤ Animation Play. In the Animation Preview dialog box, view the recording to verify that the animation is suitable for your presentation needs.

NOTE The playback of the animation gives you a general visual idea of the final output. It may not display the visual style or with the same quality of the final output.

7 If you are satisfied with the animation, click the Pause button. Then click the Save button.

8 In the Save As dialog box, choose a location and file name.

NOTE To change the file type, click the Animation Settings button. In the Animation Settings dialog box, Format option, click a file type. Click OK to return to the Save As dialog box.

9 Click Save.

Quick Reference

Commands

3DCLIP

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

3DCORBIT

Rotates the view in 3D space with continuous motion.

3DDISTANCE

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

3DFLY

Changes the 3D view in a drawing interactively to create the appearance of flying through the model.

3DFORBIT

Rotates the view in 3D space without constraining roll.

3DORBIT

Rotates the view in 3D space, but constrained to horizontal and vertical orbit only.

3DORBITCTR

Sets the center of rotation in 3D Orbit view.

3DPAN

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

3DSWIVEL

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

3DWALK

Changes the 3D view in a drawing interactively to create the appearance of walking through the model.

3DZOOM

Zooms in and out in a perspective view.

Create Motion Path Animations

Motion path animations, such as 3D animated walk-throughs of a model, allow you to visually demonstrate a model to both a technical and non-technical audience. You can record and play back a navigation to communicate your design intent dynamically.

Control a Camera Motion Path

You control the camera motion, and therefore the animation, by linking the camera and its target to a point or a path.

To create an animation using motion paths, you link the camera and its target to either a point or a path. If you want the camera to remain still, link it to a point. If you want the camera to move along a path, link it to a path.

If you want the target to remain still, link it to a point. If you want the target to move, link it to a path. You cannot link both the camera and the target to a point.

Use the same path when you want the animation view to follow in line with the camera's path. You do this by setting the target's path to None in the Motion Path Animation dialog box. This is the default setting.

NOTE To link a camera or target to a path, you must create the path object before you create the motion path animation. A path can be a line, arc, elliptical arc, circle, polyline, 3D polyline, or spline.

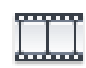
To create a motion path animation

1. In the drawing, create a path object for either the camera or the target. A path can be a line, arc, elliptical arc, circle, polyline, 3D polyline, or spline.

   NOTE The path you create is not visible in the animation.

   ---

2. If the Animations panel is not displayed on the Render tab, right-click the Render tab and click Panels ➤ Animations.

3. Click Render tab ➤ Animations panel ➤ Animation Motion Path. 

4. In the Motion Path Animation dialog box, Camera section, click either Point or Path.

5. Do one of the following:
   - To specify a new camera point, click the Pick Point button, and specify a point in the drawing. Enter a name for the point. Click OK.
   - To specify a new camera path, click the Select Path button, and specify a path in the drawing. Enter a name for the path. Click OK.
   - To specify an existing camera point or path, select it from the drop-down list.

6. In the Motion Path Animation dialog box, Target section, click either Point or Path.

7. Do one of the following:
   - To specify a new target point, click the Pick Point button, and specify a point in the drawing. Enter a name for the point. Click OK.

- To specify a new target path, click the Select Path button, and specify a path in the drawing. Enter a name for the path. Click OK.
- To specify an existing target point or path, select it from the drop-down list.

8 In the Animation Settings section, adjust the animation settings to create the animation to suit your needs.

9 When you have finished adjusting the points, paths, and settings, click Preview to view the animation, or OK to save it.

Command entry: ANIPATH

Quick Reference

Commands

ANIPATH

Saves an animation file of a camera moving or panning in a 3D model.

Specify Motion Path Settings

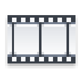
You determine the format of the animation file of a motion path animation by specifying settings in the Motion Path Animation dialog box.

Several settings control the animation's frame rate, duration, resolution, visual style, and file format.

To view a reverse motion path animation

1 If the Animations panel is not displayed on the Render tab, right-click the Render tab and click Panels ➤ Animations.

2 Click Render tab ➤ Animations panel ➤ Animation Motion Path.



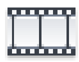
3 In the Motion Path Animation dialog box, Animation Settings section, click the Reverse check box.

4 Click OK.

Command entry: ANIPATH

To control the speed and duration of the animation

1. If the Animations panel is not displayed on the Render tab, right-click the Render tab and click Panels ➤ Animations.
2. Click Render tab ➤ Animations panel ➤ Animation Motion Path.

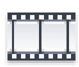
   

3. In the Motion Path Animation dialog box, Animation Settings section, specify a Frame rate (FPS).
4. Do one of the following:
   - Specify a number of frames.
   - Specify a duration, in seconds.
5. Click Preview or OK.

   ANIPATH

To set animation resolution

1. If the Animations panel is not displayed on the Render tab, right-click the Render tab and click Panels ➤ Animations.
2. Click Render tab ➤ Animations panel ➤ Animation Motion Path.

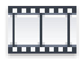
3. In the Motion Path Animation dialog box, Animation Settings section, select a resolution from the Resolution drop-down list.

Command entry: ANIPATH

To set the video format

1. If the Animations panel is not displayed on the Render tab, right-click the Render tab and click Panels ➤ Animations.

2 Click Render tab ➤ Animations panel ➤ Animation Motion Path.



3 In the Motion Path Animation dialog box, Animation Settings section, click a video format (AVI, MPG, MOV, or WMV) from the Format drop-down list.

Command entry: ANIPATH

Quick Reference

Commands

ANIPATH

Saves an animation file of a camera moving or panning in a 3D model.

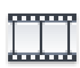
Record a Motion Path Animation

You can preview the animation before you record it and then save it in the desired format.

To preview and save a motion path animation

1 If the Animations panel is not displayed on the Render tab, right-click the Render tab and click Panels ➤ Animations.

2 Click Render tab ➤ Animations panel ➤ Animation Motion Path.



3 In the Motion Path Animations dialog box, do the following:

- Specify a point or path for the camera.
- Specify a point or path for the target.
- Adjust any animation settings.

4 To preview the animation, click the Preview button.

5 In the Animation Preview window, view the animation. When the preview animation is complete, close the Animation Preview window.

6 In the Motion Path Animations dialog box, click OK.

7 In the Save As dialog box, specify a file name and location to save the animation file.

8 Click Save.

Command entry: ANIPATH

Quick Reference

Commands

ANIPATH

Saves an animation file of a camera moving or panning in a 3D model.

Display Multiple Views in Model Space

11

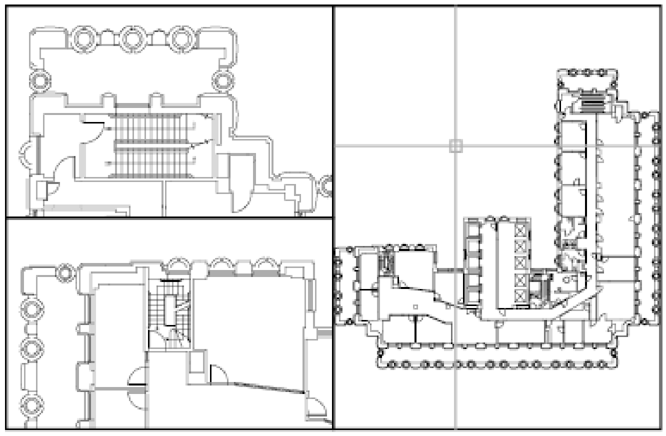
To see several views at the same time, you can split the drawing area of the Model tab into separate viewing areas called *model space viewports*. You can save arrangements of model space viewports for reuse at any time.

Set Model Space Viewports

On the Model tab, you can split the drawing area into one or more adjacent rectangular views known as *model space viewports*.

Viewports are areas that display different views of your model. As you work on the Model tab, you can split the drawing area into one or more adjacent rectangular views known as *model space viewports*. In large or complex drawings, displaying different views reduces the time needed to zoom or pan in a single view. Also, errors you might miss in one view may be apparent in the others.

Viewports created on the Model tab completely fill the drawing area and do not overlap. As you make changes in one viewport, the others are updated simultaneously. Three model space viewports are shown in the illustration.



You can also create viewports on a layout tab. You use those viewports, called *layout viewports,* to arrange the views of your drawing on a sheet. You can move and resize layout viewports. By using layout viewports, you have more control over the display; for example, you can freeze certain layers in one layout viewport without affecting the others. For more information about layouts and layout viewports, see Create Multiple-View Drawing Layouts (Paper Space) on page 431.

Use Model Space Viewports

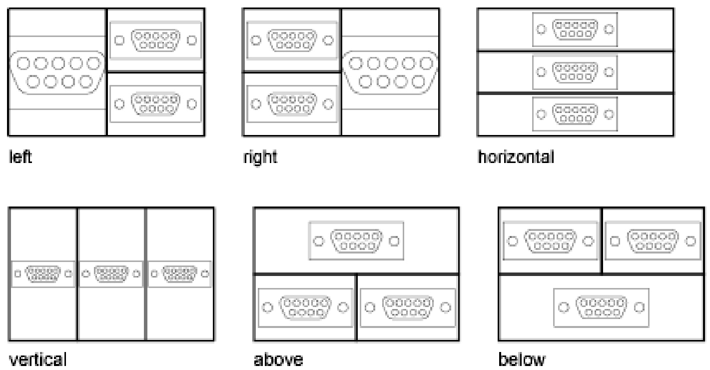
With model space viewports, you can do the following:

- Pan; zoom; set Snap, Grid, and UCS icon modes; and restore named views.
- Save user coordinate system orientations with individual viewports.
- Draw from one viewport to another when executing a command.
- Name a viewport arrangement so that you can reuse it on the Model tab or insert it on a layout tab.

Setting up different coordinate systems in individual viewports is useful if you typically work on 3D models. See Assign User Coordinate System Orientations to Viewports on page 662.

Split and Join Model Space Viewports

The illustrations below show several default model space viewport configurations.



You can easily modify model space viewports by splitting and joining them. If you want to join two viewports, they must share a common edge of the same length.

To subdivide a viewport on the Model tab

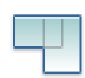
1 If you have more than one viewport, click inside the viewport you want to subdivide.

2 To indicate how many model space viewports should be created, do one of the following:
   - Click View menu ➤ Viewports ➤ 2 Viewports.
   - ClickView menu ➤ Viewports ➤ 3 Viewports.
   - Click View menu ➤ Viewports ➤ 4 Viewports

3 At the Next prompt, specify the arrangement of the new viewports.

Command entry: VPORTS

To join two viewports on the Model tab

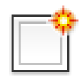
1 Click View tab ➤ Viewports panel ➤ Viewports, Join. 

2 Click within the model space viewport containing the view you want to keep.

3 Click within an adjacent viewport to join it to the first viewport.

Command entry: VPORTS

To restore a single viewport on the Model tab

- Click View tab ➤ Viewports panel ➤ New. 

Command entry: VPORTS

To switch from a layout tab to the Model tab

- Click the Model tab at the bottom of the drawing area.

Command entry: TILEMODE

Quick Reference

Commands

MODEL

Switches from a layout tab to the Model tab.

VPORTS

Creates multiple viewports in model space or paper space.

System Variables

MAXACTVP

Sets the maximum number of viewports that can be active at one time in a layout.

CTAB

Returns the name of the current (model or layout) tab in the drawing.

TILEMODE

Makes the Model tab or the last layout tab current.

Utilities

No entries

Command Modifiers

No entries

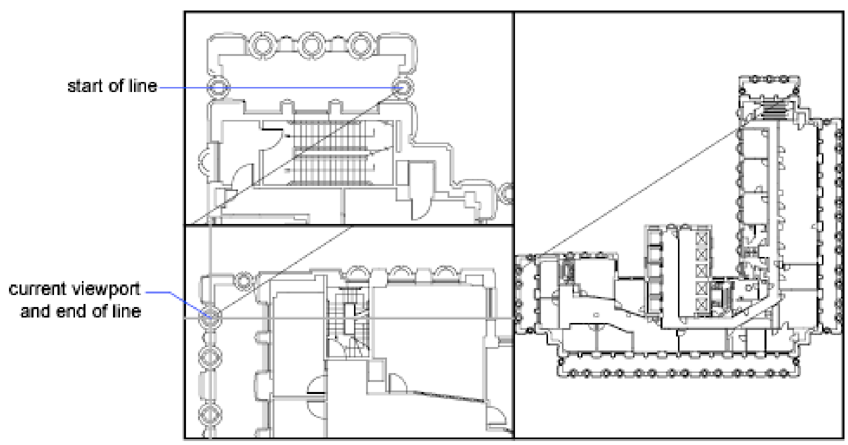
Select and Use the Current Viewport

When you use multiple viewports, one of them is the *current viewport*, which accepts cursor input and view commands.

When a viewport is current, the cursor is displayed as crosshairs rather than an arrow, and the viewport boundary is highlighted. You can change the current viewport at any time except when a View command is in progress.

To make a viewport the current viewport, you click inside it or press CTRL+R to cycle through the existing viewports.

To draw a line using two model space viewports, you start the line in the current viewport, make another viewport current by clicking within it, and then specify the endpoint of the line in the second viewport. In a large drawing, you can use this method to draw a line from a detail in one corner to a detail in a distant corner.



To make a viewport current

- Click anywhere within the viewport border.

To cycle through viewports without clicking

- Press CTRL+ R repeatedly.

Quick Reference

Commands

No entries

System Variables

CVPORT

Displays the identification number of the current viewport.

VIEWCTR

Stores the center of view in the current viewport.

VIEWSIZE

Stores the height of the view displayed in the current viewport, measured in drawing units.

Utilities

No entries

Command Modifiers

No entries

Save and Restore Model Tab Viewport Arrangements

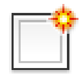
Arrangements of model viewports can be saved and restored by name.

You don't have to set up viewports and views every time you need them. With VPORTS, viewport arrangements can be saved and later restored by name. Settings that are saved with viewport arrangements include

- The number and position of viewports
- The views that the viewports contain
- The grid and snap settings for each viewport
- The UCS icon display setting for each viewport

You can list, restore, and delete the available viewport arrangements. A viewport arrangement saved on the Model tab can be inserted on a layout tab.

To save and name a viewport arrangement

1 Click View tab ➤ Viewports panel ➤ New. 

2 In the Viewports dialog box, New Viewports tab, enter a name for the viewport configuration in the New Name box.

   The name can be up to 255 characters long and contain letters, digits, the special characters dollar sign ($), hyphen (-), and underscore (_).

3 Click OK.

NOTE You can save a viewport arrangement only on the Model tab.

Command entry: VPORTS

To restore a saved viewport arrangement

1 Click View tab ➤ Viewports panel ➤ Named. 

2 In the Viewports dialog box, Named Viewports tab, select the name of the viewport configuration from the list.

3 Click OK.

Command entry: VPORTS

To delete a saved viewport arrangement

1 Click View tab ➤ Viewports panel ➤ Named. 

2 In the Viewports dialog box, Named Viewports tab, select the name of the viewport configuration you want to delete.

3 Press DELETE.

Command entry: VPORTS

To view a list of saved viewport arrangements

- Click View tab ➤ Viewports panel ➤ Named.
  The Viewports dialog box is displayed.

All saved viewport arrangements in the drawing are listed on the Named Viewports tab under Named Viewports.

Command entry: VPORTS

Quick Reference

Commands

RENAME

Changes the names assigned to items such as layers and dimension styles.

VPORTS

Creates multiple viewports in model space or paper space.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Choose a Work Process Before You Begin

Create Single-View Drawings (Model Space) 12

If you are going to create a two dimensional drawing that has one view, you can create the drawing and its annotation entirely in model space. This is the traditional method for creating drawings with AutoCAD®.

With this method, you create the building, mechanical part, or geographic area that you want to represent at full scale (1:1), but you create the text, dimensions, and the title block of the drawing at a scale to match the intended plot scale.

Quick Start for Model Space Drafting

The process of creating and plotting a drawing file in model space is very different from the process used in manual drafting.

In AutoCAD, there are two distinct working environments that are represented by the Model and layout tabs. These tabs are located near the bottom of the drawing area.

If you are going to create a two-dimensional drawing that has one view, you can create both the model and its annotation entirely in model space, not using a layout tab. This is the traditional method for creating drawings with AutoCAD. This method is simple but has several limitations, including

- It is suitable for 2D drawings only
- It does not support multiple views and view-dependent layer settings
- Scaling the annotation and title block requires computation unless you use objects.

With this method, you always draw geometric objects at full scale (1:1) and text, dimensions, and other annotation at a scale that will appear at the correct size when the drawing is plotted.

For information about using annotative objects and scaling annotations automatically, see Scale Annotations on page 1393.

See also:

- Create Multiple-View Drawing Layouts (Paper Space) on page 431
- Work with Sheets in a Sheet Set on page 479

Quick Reference

Commands

MODEL

Switches from a layout tab to the Model tab.

RENAME

Changes the names assigned to items such as layers and dimension styles.

VPORTS

Creates multiple viewports in model space or paper space.

System Variables

CVPORT

Displays the identification number of the current viewport.

MAXACTVP

Sets the maximum number of viewports that can be active at one time in a layout.

TILEMODE

Makes the Model tab or the last layout tab current.

VIEWCTR

Stores the center of view in the current viewport.

VIEWSIZE

Stores the height of the view displayed in the current viewport, measured in drawing units.

Utilities

No entries

Command Modifiers

No entries

Draw, Scale, and Annotate in Model Space

If you draw and plot from model space, you must determine and apply a scale factor to annotate objects before you plot.

You can draw and plot entirely from model space. This method is useful primarily for two-dimensional drawings that have a single view. With this method, you use the following process:

- Determine the unit of measurement (drawing units) for the drawing.
- Specify the display style for the drawing unit.
- Calculate and set the scale for dimensions, annotations, and blocks.
- Draw at full scale (1:1) in model space.
- Create the annotation and insert the blocks in model space.
- Plot the drawing at the predetermined scale.

You can also use objects if you want to scale annotations automatically. For information about using annotative objects and scaling annotations automatically, see Scale Annotations on page 1393.

Determine the Unit of Measurement

Before you begin drawing in model space, you determine the unit of measurement (drawing units) that you plan to use. You decide what each unit on the screen represents, such as an inch, a millimeter, a kilometer, or some other unit of measurement. For example, if you are drawing a motor part, you might decide that one drawing unit equals a millimeter. If you are drawing a map, you might decide that one unit equals a kilometer.

Specify the Display Style of Drawing Units

Once you have determined a drawing unit for the drawing, you need to specify the style for displaying the drawing unit, which includes the unit type and precision. For example, a value of 14.5 can be displayed as 14.500, 14-1/2, or 1'2-1/2".

Specify the display style of drawing units with the UNITS command. The default drawing unit type is decimal.

Set the Scale for Annotations and Blocks

Before you draw, you should set the scale for dimensions, annotations, and blocks in your drawings. Scaling these elements beforehand ensures that they are at the correct size when you plot the final drawing.

You should enter the scale for the following objects:

- **Text.** Set the text height as you create text or by setting a fixed text height in the text style (STYLE).
- **Dimensions.** Set the dimension scale in a dimension style (DIMSTYLE) or with the DIMSCALE system variable.
- **Linetypes.** Set the scale for noncontinuous linetypes with the CELTSCALE and LTSCALE system variables.
- **Hatch patterns.** Set the scale for hatch patterns in the Hatch and Gradient dialog box (HATCH) or with the HPSCALE system variable.
- **Blocks.** Specify the insertion scale for blocks either as you insert them, or set an insertion scale in the Insert dialog box (INSERT) or in DesignCenter (ADCENTER). The system variables used for inserting blocks are INSUNITS, INSUNITSDEFSOURCE, and INSUNITSDEFTARGET. This also applies to the border and title block of the drawing.

You can also use objects if you want to scale annotations automatically. For information about using annotative objects and scaling annotations automatically, see Scale Annotations on page 1393.

Determine the Scale Factor for Plotting

To plot your drawing from the Model tab, you calculate the exact scale factor by converting the drawing scale to a ratio of 1:*n*. This ratio compares plotted units to drawing units that represent the actual size of the objects you are drawing.

For example, if you plan to plot at a scale of 1/4 inch = 1 foot, you would calculate the scale factor 48 as follows:

1/4" = 12"

1 = 12 x 4

1 (plotted unit) = 48 (drawing units)

Using the same calculation, the scale factor for 1 centimeter = 1 meter is 100, and the scale factor for 1 inch = 20 feet is 240.

Sample Scale Ratios

The sample architectural scale ratios in the table can be used to calculate text sizes in model space.

Scale	Scale factor	To plot text size at	Set drawing text size to
1 cm = 1 m	100	3 mm	30 cm
1/8" = 1'-0"	96	1/8"	12"
3/16" = 1'-0"	64	1/8"	8"
1/4" = 1'-0"	48	1/8"	6"
3/8" = 1'-0"	32	1/8"	4"
1/2" = 1'-0"	24	1/8"	3"
3/4" = 1'-0"	16	1/8"	2"
1" = 1'-0"	12	1/8"	1.5"
1 1/2" = 1'-0"	8	1/8"	1.0"

If you are working in metric units, you might have a sheet size of 210 x 297 mm (A4 size) and a scale factor of 20. You calculate grid limits as follows:

210 x 20 = 4200 mm

297 x 20 = 5900 mm

See also:

- Specify Units and Unit Formats on page 180

To specify the display style for drawing units

1. Click Format menu ➤ Units. 

2. In the Drawing Units dialog box, set the unit values for your drawing.
3. As you change unit settings, you can see examples under Sample Output.
   - Under Length, select a unit type and level of precision. This determines the display style for linear drawing units.
   - Under Drawing Units for DesignCenter Blocks, select the unit that you want used to scale blocks, images, or other content inserted into the drawing. If you do not want inserted content to be scaled, select Unitless.
   - Under Angle, select an angle type and precision. This determines the display style for angular drawing units. The default starting angle, 0 degrees, is toward 3 o'clock (or east).
   - To specify an angle direction, click Direction, and then select the base angle in the Direction Control dialog box. The angle direction controls the point from which angles are measured and the direction in which they are measured. If you select Other, you can enter an angle, or click Angle to specify an angle using your pointing device. The default positive angle measurement is counterclockwise.
4. Click OK to exit each dialog box.

Command entry: UNITS

Quick Reference

Commands

ADCENTER

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

DIMSTYLE

Creates and modifies dimension styles.

INSERT

Inserts a block or drawing into the current drawing.

LINETYPE

Loads, sets, and modifies linetypes.

PLOT

Plots a drawing to a plotter, printer, or file.

STYLE

Creates, modifies, or specifies text styles.

UNITS

Controls coordinate and angle display formats and precision.

System Variables

CELTSCALE

Sets the current object linetype scaling factor.

DIMSCALE

Sets the overall scale factor applied to dimensioning variables that specify sizes, distances, or offsets.

HPSCALE

Specifies the hatch pattern scale factor, which must be greater than zero.

HPSPACE

Specifies the hatch pattern line spacing for user-defined simple patterns, which must be greater than zero.

INSUNITS

Specifies a drawing-units value for automatic scaling of blocks, images, or xrefs when inserted or attached to a drawing.

INSUNITSDEFSOURCE

Sets source content units value when INSUNITS is set to 0.

INSUNITSDEFTARGET

Sets target drawing units value when INSUNITS is set to 0.

LTSCALE

Sets the global linetype scale factor.

LUNITS

Sets linear units.

TEXTSIZE

Sets the default height for new text objects drawn with the current text style.

Utilities

No entries

Command Modifiers

No entries

Create Multiple-View Drawing Layouts (Paper Space)

13

Paper space is a sheet layout environment where you can specify the size of your sheet, add a title block, display multiple views of your model, and create dimensions and notes for your drawing.

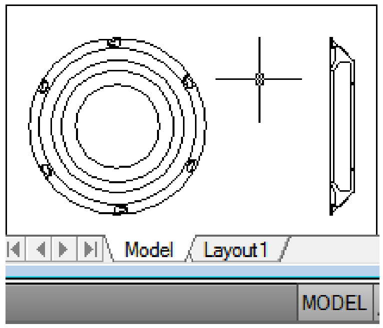
Quick Start for Layouts

There are two distinct working environments, or "spaces," in which you can create objects in a drawing. These are represented by the Model and layout tabs.

Typically, a model composed of geometric objects is created in a three-dimensional space called *model space*. A final layout of specific views and annotations of this model is created in a two-dimensional space called *paper space*. These spaces are accessible on two or more tabs near the bottom of the drawing area: the Model tab and one or more layout tabs.

NOTE These tabs can be hidden, appearing instead as buttons on the status bar at the bottom-center of the application window.

Working on the Model tab, you draw a model of your subject at 1:1 scale. Working on a layout tab, you can create one or more *layout viewports*, dimensions, notes, and a title block to represent a drawing sheet.



Each layout viewport is like a picture frame containing a “photograph” of the model in model space. Each layout viewport contains a view that displays the model at the scale and orientation that you specify. You can also specify which layers are visible in each layout viewport.

After you finish arranging the layout, you turn off the layer that contains the layout viewport objects. The views are still visible, and you can plot the layout without displaying the viewport boundaries.

Quick Reference

Commands

LAYOUT

Creates and modifies drawing layout tabs.

LAYOUTWIZARD

Creates a new layout tab and specifies page and plot settings.

MODEL

Switches from a layout tab to the Model tab.

MSPACE

Switches from paper space to a model space viewport.

MVIEW

Creates and controls layout viewports.

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PSETUPIN

Imports a user-defined page setup into a new drawing layout.

PSPACE

Switches from a model space viewport to paper space.

VPORTS

Creates multiple viewports in model space or paper space.

VPLAYER

Sets layer visibility within viewports.

System Variables

MAXACTVP

Sets the maximum number of viewports that can be active at one time in a layout.

PSLTSCALE

Controls the linetype scaling of objects displayed in paper space viewports.

TILEMODE

Makes the Model tab or the last layout tab current.

Utilities

No entries

Command Modifiers

No entries

Understand the Layout Process

When you use a layout tab to prepare your drawing for plotting, you follow a series of steps in a process.

You design the subject of your drawing on the Model tab (in model space) and prepare it for plotting on a layout tab (in paper space).

There is one Model tab and one or more layout tabs at the bottom of the drawing window.

NOTE These tabs can be hidden, appearing instead as buttons on the status bar at the bottom-center of the application window.

You can initialize a layout by clicking on its tab to activate the previously unused layout. A layout does not contain any plot settings before initialization. Once initialized, layouts can be drawn upon, published, and added to sheet sets as sheets (after the drawing has been saved).

Process Summary

When you prepare a layout, you typically step through the following process:

- Create a model of your subject on the Model tab.
- Click a layout tab.
- Specify layout page settings such as plotting device, paper size, plot area, plot scale, and drawing orientation.
- Insert a title block into the layout (unless you have started with a drawing template that already has a title block).
- Create a new layer to be used for layout viewports.
- Create layout viewports and position them on the layout.
- Set the orientation, scale, and layer visibility of the view in each layout viewport.
- Add dimensions and annotate in the layout as needed.
- Turn off the layer containing the layout viewports.
- Plot your layout.

You can also use objects if you want to annotate your drawing in model space and scale the annotations automatically. For information about using annotative objects and scaling annotations automatically, see Scale Annotations on page 1393.

The other topics in this chapter provide additional detail on how to create, use, and modify layouts and layout viewports.

Quick Reference

Commands

LAYOUT

Creates and modifies drawing layout tabs.

LAYOUTWIZARD

Creates a new layout tab and specifies page and plot settings.

MODEL

Switches from a layout tab to the Model tab.

MSPACE

Switches from paper space to a model space viewport.

MVIEW

Creates and controls layout viewports.

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PLOT

Plots a drawing to a plotter, printer, or file.

PSETUPIN

Imports a user-defined page setup into a new drawing layout.

PSPACE

Switches from a model space viewport to paper space.

VPLAYER

Sets layer visibility within viewports.

VPMAX

Expands the current layout viewport for editing.

VPMIN

Restores the current layout viewport.

VPORTS

Creates multiple viewports in model space or paper space.

System Variables

LAYOUTREGENCTL

Specifies how the display list is updated in the Model tab and layout tabs.

MAXACTVP

Sets the maximum number of viewports that can be active at one time in a layout.

TILEMODE

Makes the Model tab or the last layout tab current.

Utilities

No entries

Command Modifiers

No entries

Work with Model Space and Paper Space

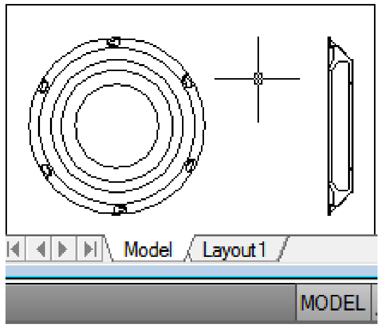
There are several benefits to switching between model space and paper space to perform certain tasks. Use model space for creating and editing your model. Use paper space for composing your drawing sheet and defining views.

Work on the Model Tab

The Model tab accesses a limitless drawing area called *model space*. In model space, you draw, view, and edit your model.

In model space, you draw your model at 1:1 scale, and you decide whether one unit represents one millimeter, one centimeter, one inch, one foot, or whatever unit is most convenient or customary in your business.

On the Model tab, you can view and edit model space objects. The crosshairs cursor is active over the entire drawing area.



In model space, you can also define named views that you display in layout viewports on a layout.

To activate the Model tab

Do one of the following to make the Model tab current:

- Click the Model tab.
- Right-click any layout tab or the Model tab. Click Activate Model Tab.
- If the Model and layout tabs are hidden, click the Model button on the status bar at the bottom of the application window.

Quick Reference

Commands

MODEL

Switches from a layout tab to the Model tab.

MSPACE

Switches from paper space to a model space viewport.

PSPACE

Switches from a model space viewport to paper space.

System Variables

LAYOUTREGENCTL

Specifies how the display list is updated in the Model tab and layout tabs.

MAXACTVP

Sets the maximum number of viewports that can be active at one time in a layout.

TILEMODE

Makes the Model tab or the last layout tab current.

Utilities

No entries

Command Modifiers

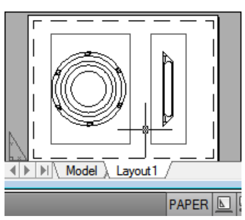
No entries

Work on a Layout Tab

Layout tabs access an area called *paper space*. In paper space, you place your title block, create layout viewports to display views, dimension your drawing, and add notes.

In paper space, one unit represents the paper distance on a plotted sheet. The units will be in either millimeters or inches, depending on the plot setup for your plotter.

On a layout tab, you can view and edit paper space objects, such as layout viewports and title blocks. You can also move an object (such as a leader or a title block) from model space to paper space (or vice versa). The crosshairs cursor is active over the entire layout area.



NOTE These tabs can be hidden, appearing instead as buttons on the status bar at the bottom-center of the application window. To display the tabs, right-click the Model or layout button and click Display Layout and Model Tabs on the shortcut menu.

Create Additional Layout Tabs

By default, a new drawing starts with two layout tabs, named Layout1 and Layout2. If you use a drawing template or open an existing drawing, the layout tabs in your drawing may be named differently.

You can create a new layout tab using one of the following methods:

- Add a new layout tab with no settings and then specify the settings in the Page Setup Manager.
- Use the Create Layout wizard to create the layout tab and specify settings.
- Copy a layout tab and its settings from the current drawing file.
- Import a layout tab from an existing drawing template (DWT) file or drawing (DWG) file.

NOTE You can create multiple layouts in a drawing; each layout can contain different plot settings and paper sizes. However, to avoid confusion when transmitting and publishing drawings, it is usually recommended that you create only one layout for each drawing.

Use the Layout Wizard to Specify Layout Settings

You can create a new layout using the Create Layout wizard. The wizard prompts you for information about the layout settings, including

- A name for the new layout
- The printer associated with the layout
- A paper size to use for the layout
- The orientation of the drawing on the paper
- A title block
- Viewport setup information
- A location for the viewport configuration in the layout

You can edit the information entered in the wizard later. Click Output tab ➤ Plot panel ➤ Page Setup Manager.

To specify the page setup for the current layout

- Click Output tab ➤ Plot panel ➤ Page Setup Manager.

- Right-click the current layout tab. Click Page Setup.

Toolbar: Layouts 
Command entry: PAGESETUP

To create a new layout

1. Click Insert menu ➤ Layout ➤ New Layout.
2. Enter the name of the new layout at the command prompt.
   A new layout tab is created. To switch to the new layout, choose the layout tab.

Alternatively, you can right-click the existing layout tab and click New layout. To rename a layout, double-click the layout tab.

Toolbar: Layouts 
Command entry: LAYOUT

To import a layout from a template

1. Click Insert menu ➤ Layout ➤ Layout From Template.
2. In the Select File dialog box, select a DWT or DWG file to import a layout from.
3. Click Open.
4. In the Insert Layout(s) dialog box, select a layout to import.
   A new layout tab is created. To switch to the new layout, click the layout tab.

Toolbar: Layouts 
Command entry: LAYOUT

To create a layout using a wizard

1. Click Insert menu ➤ Layout ➤ Layout Wizard.

2 On each page of the Create Layout wizard, select the appropriate settings for the new layout.

   When finished, the new layout will be the current layout tab.

Command entry: LAYOUTWIZARD

To duplicate a layout

1 Right-click the layout tab of the layout you want to duplicate. Click Move or Copy.

2 In the Move or Copy dialog box, select a position for the new layout tab.

3 Make sure that Create a Copy is selected.

4 Click OK.

Alternatively, you can press CTRL+drag to create a duplicate copy of the selected layout or a group of selected layouts.

NOTE The Model tab cannot be duplicated.

Command entry: LAYOUT

To rename a layout

1 Right-click the layout tab for the layout you want to rename. Click Rename.

2 In the Rename Layout dialog box, enter the new name for the layout.

3 Click OK.

Alternatively, you can double-click the layout tab for the layout you want to rename and directly enter the new name for the layout.

NOTE The Model tab cannot be renamed.

Command entry: LAYOUT

To delete a layout

1 Right-click the layout tab for the layout you want to delete. Click Delete.

2 In the warning box, click OK to delete the layout.

Any named views associated with the layout are deleted automatically when the layout is deleted.

NOTE The Model tab cannot be deleted.

Command entry: LAYOUT

To rearrange layout tabs

1 Right-click the layout tab of the layout you want to reposition. Click Move or Copy.

2 In the Move or Copy dialog box, select the layout tab that should be placed immediately after the moved or copied layout tab. If you want to move the layout tab to the end of the list of layout tabs, select Move to End.

3 Click OK.

Alternatively, you can click and drag the layout tab of the layout you want to reposition and place the layout at the desired position.

NOTE The Model tab cannot be repositioned.

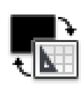
Command entry: LAYOUT

To make a layout current

Do one of the following to make a layout current:

- Click the layout tab for the layout that you want to make current.
- Press CTRL+Page Down to cycle through the layout tabs from left to right or CTRL+Page Up to cycle through the layout tabs from right to left. Stop when you get to the tab of the layout that you want to make current.

To move an object from model space to paper space (or vice versa)

1 Click Modify menu ➤ Change Space. 

2 Select one or more objects to move.

3 Press ENTER to end the command.

Command entry: CHSPACE

To activate the previous layout

1 Right-click any layout tab or the Model tab.

2 Click Activate Previous Layout.

To select all layouts

■ Right-click any layout tab. Click Select All Layouts.

Command entry: LAYOUT

To plot a layout

1 Click Output tab ➤ Plot panel ➤ Plot.

2 Do *one* of the following:
   - Right-click a layout tab. Click Plot.
   - Hold down SHIFT to select several layout tabs. Right-click a layout tab. Click Publish Selected Layouts.

3 Select settings in the Plot or the Publish dialog box. Click OK or Publish.

Toolbar: Standard 
Command entry: PLOT

Quick Reference

Commands

LAYOUT

Creates and modifies drawing layout tabs.

LAYOUTWIZARD

Creates a new layout tab and specifies page and plot settings.

MODEL

Switches from a layout tab to the Model tab.

MSPACE

Switches from paper space to a model space viewport.

MVIEW

Creates and controls layout viewports.

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PLOT

Plots a drawing to a plotter, printer, or file.

PSETUPIN

Imports a user-defined page setup into a new drawing layout.

PSPACE

Switches from a model space viewport to paper space.

VPORTS

Creates multiple viewports in model space or paper space.

VPLAYER

Sets layer visibility within viewports.

System Variables

CTAB

Returns the name of the current (model or layout) tab in the drawing.

CVPORT

Displays the identification number of the current viewport.

LAYOUTREGENCTL

Specifies how the display list is updated in the Model tab and layout tabs.

MAXACTVP

Sets the maximum number of viewports that can be active at one time in a layout.

PLOTROTMODE

Controls the orientation of plots.

TILEMODE

Makes the Model tab or the last layout tab current.

Utilities

No entries

Command Modifiers

No entries

Access Model Space from a Layout Viewport

You can access model space from a layout viewport to edit objects, to freeze and thaw layers, and to adjust the view.

After creating viewport objects, you can access model space from a layout viewport to perform the following tasks:

- Create and modify objects in model space inside the layout viewport.
- Pan the view inside the layout viewport and change layer visibility.

The method you use to access model space depends on what you plan to do.

Create and Modify Objects in a Layout Viewport

If you plan to create or modify objects, use the button on the status bar to maximize the layout viewport. The maximized layout viewport expands to fill the drawing area. The center point and the layer visibility settings of the viewport are retained, and the surrounding objects are displayed.

You can pan and zoom while you are working in model space, but when you restore the viewport to return to paper space, the position and scale of the objects in the layout viewport are restored.

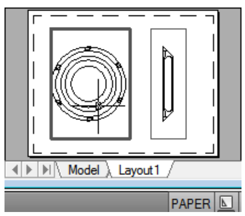
NOTE If you use PLOT while a viewport is maximized, the layout tab is restored before the Plot dialog box is displayed. If you save and close the drawing while a viewport is maximized, the drawing opens with the layout tab restored.

If you choose to switch to the Model tab to make changes, the layer visibility settings are the settings for the drawing as a whole, not the settings for that particular layout viewport. Also, the view is not centered or magnified the same way it is in the layout viewport.

Adjust the View in a Layout Viewport

If you plan to pan the view and change the visibility of layers, double-click within a layout viewport to access model space. The viewport border becomes thicker, and the crosshairs cursor is visible in the current viewport only. All active viewports in the layout remain visible while you work. You can freeze and thaw layers in the current viewport in the Layer Properties Manager, and you can pan the view. To return to paper space, double-click an empty area on the layout outside a viewport. The changes you made are displayed in the viewport.

If you set the scale in the layout viewport before you access model space, you can lock the scale to prevent changes. When the scale is locked, you cannot use ZOOM while you work in model space.



To switch between model space and paper space on a layout

On a layout, use one of the following methods:

- If you are in paper space, double-click within a layout viewport.
  You are now in model space. The selected layout viewport becomes the current viewport, and you can pan the view and change layer properties. If you need to make significant changes to the model, it is recommended that you use VPMAX to maximize the layout viewport or switch to the Model tab.
- If you are in model space in a layout viewport, double-click outside the viewport.
  You are now in paper space. You can create and modify objects on the layout.
- If you are in model space and want to switch to another layout viewport, double-click within another layout viewport, or press CTRL+R to cycle through the existing layout viewports.

To edit in a maximized layout viewport

1 Click the boundary of the layout viewport to select it.

NOTE You can maximize a locked viewport and modify objects. When you restore the viewport, it is locked again.

2 On the status bar, click the Maximize Viewport button.

   You can restore the viewport and maximize another viewport by clicking one of the arrows next to the Maximize Viewport button.

3 Make any changes.

4 To return to the layout viewport, click the Restore Viewport button on the status bar.

   The center point and magnification are returned to the settings that were in effect before the viewport was maximized.

Command entry: VPMAX, VPMIN
Shortcut menu: Right-click. Click Maximize Viewport.

Quick Reference

Commands

MODEL

Switches from a layout tab to the Model tab.

MSPACE

Switches from paper space to a model space viewport.

PSPACE

Switches from a model space viewport to paper space.

VPMAX

Expands the current layout viewport for editing.

VPMIN

Restores the current layout viewport.

System Variables

VPMAXIMIZEDSTATE

Indicates whether the viewport is maximized or not.

Utilities

No entries

Command Modifiers

No entries

Export a Layout to Model Space

You can export all visible objects from the current layout to model space.

You can export all visible objects from the current layout to the model space with the EXPORTLAYOUT command. Objects that are outside the boundaries of “paper” in the layout are also exported.

Some objects are not exported to the model space drawing. The objects are

- Materials
- Cameras
- Lights
- Named views
- Objects on layers that are disabled (off) or frozen
- Model space objects not visible in a given viewport

Changes to Exported Objects

When exported, some objects become a different object type, or are modified in order to maximize visual fidelity with the layout.

Object Type	Representation in Exported Drawing
Dimensions	Dimensions that exceed the boundaries of the layout viewport are exploded.
Standard or dynamic block (with or without attributes)	Standard or dynamic blocks, with or without attributes, that exceed the boundaries of the layout viewport will be converted to a new, anonymous block. Attributes are converted to text objects in the block.

Object Type	Representation in Exported Drawing
	NOTE The “Allow Exploding” setting (a setting on the block definition) is ignored if the block exceeds the boundaries of the layout viewport.
Annotative objects	Objects become non-annotative.
External reference (xref)	An Xref with nested objects that exceed the boundaries of the layout viewport is converted to a block reference and exploded.
Layout viewport	Layout viewports are represented by either a polyline or the clipped viewport object.
Custom objects	Custom objects are exploded and converted to anonymous blocks.
Xref clipped with the XCLIP command	Xrefs clipped with the XCLIP command are converted to a clipped block reference.
Visual Styles	The 2D wireframe visual style is used.
Perspective viewports	Objects within a perspective viewport will have parallel projection.

NOTE Objects that can be directly trimmed without changing their types are not listed on the table.

Visual Changes to Objects

Not all objects when displayed in a layout will display the same in the Model tab of the exported drawing. This includes (but is not limited to) the following instances:

- The same object displayed in multiple viewports becomes multiple objects in the exported model space drawing. In addition, objects are transformed and often scaled. Both of these can affect data extraction of blocks.
- Some objects are converted or exploded in order to trim them.
- Viewports in a layout can each have a different visual style; only a single visual style can be used in model space.

- Polylines with width that is clipped by the viewport boundary may not be accurately trimmed in the exported drawing.

Recommendations

When exporting a layout to model space, consider the following:

- The performance of the EXPORTLAYOUT command may be slower if a model space viewport is active.
- In the exported drawing, the viewport displays the original linetype, which may not match the look of the original drawing. If this happens, assign "Continuous" linetype to viewports in the original drawing.
- Linetype scaling may not be accurately maintained for objects in xrefs and blocks if PSLTSCALE is 0.
- If you have a problem with xrefs during export, detach unresolved xrefs or manually bind xrefs and then use the EXPORTLAYOUT command.
- Superhatch objects (from Express Tools) are exported, but the hatching may not stay within the original boundaries. You can use the TRIM command in the exported drawing to correct any problems with visual appearance.

To export a layout to a model space drawing

1 Click File menu ➤ Export Layout to Model. 

NOTE You can use the EXPORTLAYOUT command only from a layout tab.

2 In the Export Layout to Model Space Drawing dialog box, enter a file name.

3 Specify the location where you want to save the file. The default location is the location of the current drawing.

4 Click Save.

Command entry: EXPORTLAYOUT

To achieve better visual fidelity for a drawing containing AEC objects

1 In an open drawing, enter **aectoacad**. The AECTOACAD command creates a new DWG file with all AEC objects exploded into basic AutoCAD objects.

   NOTE It is also recommended to select Insert as the bind type when using the AECTOACAD command.

2 Navigate to the converted drawing. Click Open.

3 On the command line, enter **exportlatyout**.

Command entry: EXPORTLAYOUT

Quick Reference

Commands

EXPORTLAYOUT

Creates a visual representation of the current layout in the model space of a new drawing.

System Variables

No entries

Utilities

No entries

Command Modifiers

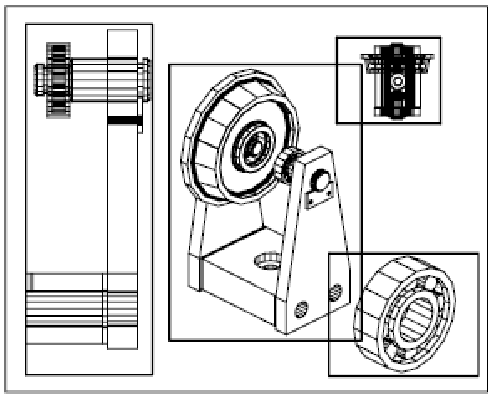
No entries

Create and Modify Layout Viewports

You can create a single layout viewport that fits the entire layout or create multiple layout viewports in the layout. Once you create the viewports, you can change their size, their properties, and also scale and move them as needed.

NOTE It is important to create layout viewports on their own layer. When you are ready to plot, you can turn off the layer and plot the layout without plotting the boundaries of the layout viewports.

With MVIEW, you have several options for creating one or more layout viewports. You can also use COPY and ARRAY to create multiple layout viewports.



Create Nonrectangular Layout Viewports

You can create a new viewport with nonrectangular boundaries by converting an object drawn in paper space into a layout viewport.

You can use the MVIEW command to create nonrectangular viewports.

- With the Object option, you can select a closed object, such as a circle or closed polyline created in paper space, to convert into a layout viewport. The object that defines the viewport boundary is associated with the viewport after the viewport is created

- With the Polygonal option, you can create a nonrectangular layout viewport by specifying points. The prompts are the same as the prompts for creating a polyline

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NOTE When you want to suppress the display of the boundary of a layout viewport, you should turn off the layer of the nonrectangular viewport instead of freezing it. If the layer of a nonrectangular layout viewport is frozen, the viewport is not clipped correctly.

Redefine Layout Viewport Boundaries

You can redefine the boundary of a layout viewport by using the VPCLIP command. You can either select an existing object to designate as the new boundary, or specify the points of a new boundary. The new boundary does not clip the old boundary, it redefines it.

A nonrectangular viewport consists of two objects: the viewport itself and the clipping boundary. You can make changes to the viewport, the clipping boundary, or both.

NOTE In the Properties palette, the default selection for a nonrectangular viewport is Viewport. This is because you are more likely to change the properties of the viewport than of the clipping boundary.

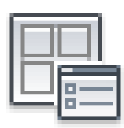
Resize Layout Viewports

If you want to change the shape or size of a layout viewport, you can use grips to edit the vertices just as you edit any object with grips.

To create a new layout viewport

1. On the layout tab, Click View tab ➤ Viewports panel ➤ New.
2. In the Viewports dialog box, New Viewports tab, under Standard Viewports, select Single.
3. Click to specify one corner of the new layout viewport.
4. Click to specify the opposite corner.

   A new layout viewport object is available and displays a default view. To adjust the view, double-click within the layout viewport to access model space.

Toolbar: Viewports 
Command entry: VPORTS, MVIEW

To create a viewport configuration on a layout

1. Click a layout tab.
2. On the layout tab, Click View tab ➤ Viewports panel ➤ New.
3. In the Viewports dialog box, New Viewports tab, under standard viewports select a viewport configuration from the list.
4. Under Setup, select either 2D or 3D.

When you select 3D, a set of standard 3D views is applied to each viewport in the configuration.

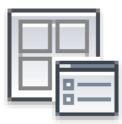
5 Under Viewport Spacing, select the amount of spacing you want to add between the viewports.

6 To change a view, select a viewport in the preview image. Under Change View To, select a view from the list of standard views.

The list includes top, bottom, front, back, left, right, and isometric views, along with any named views that are saved in the drawing. The selected view is displayed under Preview.

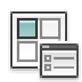
7 Click OK.

8 In the drawing area, specify two points to indicate the area to contain the viewport configuration.

Toolbar: Viewports 
Command entry: VPORTS

To place a named viewport configuration into a layout

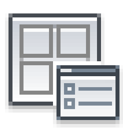
1 Click a layout tab.

2 Click View tab ➤ Viewports panel ➤ Named. 

3 In the Viewports dialog box, Named Viewports tab, select the named viewport configuration from the list.

4 Click OK.

5 In the layout, specify a location for the named viewport configuration.

Toolbar: Viewports
Command entry: VPORTS 

To modify layout viewport properties using the Properties palette

1 Click the border of the layout viewport whose properties you want to modify.

2 Click View tab ➤ Palettes panel ➤ Properties

3 In the Properties palette, select the value for the property you want to modify. Enter a new value or select a new setting from the list provided.

   The new property setting or value is assigned to the selected layout viewport.

Toolbar: Standard
Command entry: PROPERTIES
Shortcut menu: Select the viewport and right-click in the drawing area. Click Properties.

To clip a layout viewport boundary

1 Click View tab ➤ Viewports panel ➤ Viewport Clip.

2 Select the viewport to clip.

3 (Optional) Enter **d** (Delete) to delete an existing clipping boundary.

4 Do one of the following:

5 ■ Enter **p** (Polygonal) to specify a series of points to define a polygonal boundary.

   ■ Select a paper space object that will define the new viewport boundary.

Command entry: Select the viewport to clip and right-click in the drawing area. Click Viewport Clip.

Quick Reference

Commands

MVIEW

Creates and controls layout viewports.

PROPERTIES

Controls properties of existing objects.

VPCLIP

Clips layout viewport objects and reshapes the viewport border.

VPORTS

Creates multiple viewports in model space or paper space.

System Variables

MAXACTVP

Sets the maximum number of viewports that can be active at one time in a layout.

Utilities

No entries

Command Modifiers

No entries

Control Views in Layout Viewports

When you create a layout, you can add layout viewports that act as windows into model space. In each layout viewport, you can control the view that is displayed.

Scale Views in Layout Viewports

To scale each displayed view in the plotted drawing accurately, set the scale of each view relative to paper space.

You can change the view scale of the viewport using

- The Properties palette
- The XP option of the ZOOM command
- The Viewports toolbar

NOTE You can modify the list of scales that are displayed in all view and plot scale lists with SCALELISTEDIT.

When you work in a layout, the scale factor of a view in a layout viewport represents a ratio between the actual size of the model displayed in the viewport and the size of the layout. The ratio is determined by dividing the paper space units by the model space units. For example, for a quarter-scale drawing, the ratio would be a scale factor of one paper space unit to four model space units, or 1:4.

Scaling or stretching the layout viewport border does not change the scale of the view within the viewport.

Lock the Scale of Layout Viewports

Once you set the viewport scale, you cannot zoom within a viewport without changing the viewport scale. By locking the viewport scale first, you can zoom in to view different levels of detail in your viewport without altering the viewport scale.

Scale locking locks the scale that you set for the selected viewport. Once the scale is locked, you can continue to modify the geometry in the viewport without affecting the viewport scale. If you turn a viewport's scale locking on, most of the viewing commands, such as VPOINT, DVIEW, 3DORBIT, PLAN, and VIEW, no longer function in that viewport.

NOTE Viewport scale locking is also available for nonrectangular viewports. To lock a nonrectangular viewport, you must perform an extra step in the Properties palette to select the viewport object rather than the viewport clipping boundary.

To modify a layout viewport scale using the Properties palette

1. Make sure you are on a layout tab in paper space.
2. Click the border of the viewport whose scale you want to modify.
3. Right-click, and then click Properties.
4. In the Properties palette, select Standard Scale, and then select a new scale from the list.

   The scale you choose is applied to the viewport.

   NOTE To use a custom scale, enter a scale in the Custom Scale field in the Properties palette.

Toolbar: Standard 
Command entry: PROPERTIES
Shortcut menu: Select the viewport and right-click in the drawing area. Click Properties.

To turn on scale locking in a layout viewport

1. In the layout, click the viewport whose scale you want to lock.
2. If necessary, open the Properties palette.
3. In the Properties palette, do one of the following:
   - If you selected a rectangular viewport, select Display Locked, and then click Yes.
   - If you selected a nonrectangular viewport, first click All (2) and select Viewport (1). Then select Display Locked, and click Yes.

   The current viewport's scale is locked. If you change the zoom factor in the viewport, only paper space objects are affected.

Toolbar: Standard 
Command entry: PROPERTIES
Shortcut menu: Select the viewport and right-click in the drawing area. Click Properties.

Quick Reference

Commands

SCALELISTEDIT

Controls the list of scales available for layout viewports, page layouts, and plotting.

MVIEW

Creates and controls layout viewports.

PROPERTIES

Controls properties of existing objects.

VPORTS

Creates multiple viewports in model space or paper space.

ZOOM

Increases or decreases the magnification of the view in the current viewport.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

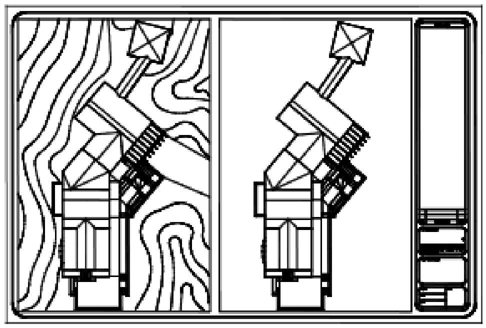
Control Visibility in Layout Viewports

You can control the visibility of objects in layout viewports using several methods. These methods are useful for emphasizing or hiding different elements of a drawing, and for reducing screen regeneration time.

Freeze Specified Layers in a Layout Viewport

A major benefit to using layout viewports is that you can selectively freeze layers in each layout viewport. You can also specify default visibility settings for new viewports and for new layers. As a result, you can view different objects in each layout viewport.

You can freeze or thaw layers in current and future layout viewports without affecting other viewports. Frozen layers are invisible. They are not regenerated or plotted. In the illustration, the layer showing terrain has been frozen in one viewport.



Thawing the layer restores visibility. The easiest way to freeze or thaw layers in the current viewport is to use the Layer Properties Manager.

In the Layer Properties Manager, on the right side, use the column labeled VP Freeze to freeze one or more layers in the current layout viewport. To display the VP Freeze column, you must be on a layout tab. Specify the current layout viewport by double-clicking anywhere within its borders.

Freeze or Thaw Layers Automatically in New Layout Viewports

You can set visibility defaults for specific layers in all new layout viewports. For example, you can restrict the display of dimensions by freezing the DIMENSIONS layer in all new viewports. If you create a viewport that requires dimensions, you can override the default setting by changing the setting in the current viewport. Changing the default for new viewports does not affect existing viewports.

Create New Layers That Are Frozen in All Layout Viewports

You can create new layers that are frozen in all existing and new layout viewports. Then you can thaw the layers in the viewports you specify. This is a shortcut for creating a new layer that is visible only in a single viewport.

To freeze or thaw layers in the current layout viewport

1. Double-click within a layout viewport to make it current.
2. Click Home tab ➤ Layer panel ➤ Layer Properties.
3. In the Layer Properties Manager, select the layers to freeze or thaw.

   Hold down CTRL to select more than one layer. Hold down SHIFT to select a sequence of layers.
4. Click the icon in the VP Freeze column for one of the selected layers.

5 Click OK.

Toolbar: Layers

Command entry: LAYER

To view a list of layers that are frozen in the current viewport

1 Click a layout tab.

2 Double-click within a layout viewport to make it current.

3 Click Home tab ➤ Layer panel ➤ Layer Properties.

4 In the Layer Properties Manager, look at the VP Freeze column for the Freeze/Thaw in Current Viewports icon.

5 Click OK.

Toolbar: Layers

Command entry: LAYER, VPLAYER

To freeze or thaw layers in all viewports

1 Click a layout tab.

2 Click Home tab ➤ Layer panel ➤ Layer Properties.

3 In the Layer Properties Manager, select one or more layers to freeze or thaw.

Hold down CTRL to select more than one layer. Hold down SHIFT to select a sequence of layers.

4 In the Freeze column, click the icon to freeze or thaw.

Toolbar: Layers

Command entry: LAYER

To freeze or thaw layers in paper space

1 Click a layout tab.

2 Make sure that you are in paper space. (On the status bar, PAPER is on.)

3 Click Home tab ➤ Layer panel ➤ Layer Properties.

4 In the Layer Properties Manager, select the layer or layers you want to freeze or thaw.

5 In the Freeze column, click the icon to change a layer's state. The sun icon means that a layer is thawed; the snowflake icon means that a layer is frozen.

6 Click OK.

Toolbar: Layers
Command entry: LAYER

To freeze or thaw layers in all new viewports

1 Click a layout tab.

2 Click Home tab ➤ Layer panel ➤ Layer Properties.

3 In the Layer Properties Manager, select the layer or layers you want to be automatically frozen or thawed in new viewports you create.

   Hold down CTRL to select more than one layer. Hold down SHIFT to select a sequence of layers.

4 In the New VP Freeze column, click the icon to change a layer's state. The sun icon means that a layer is thawed; the snowflake icon means that a layer is frozen.

5 Click OK.

Toolbar: Layers
Command entry: LAYER

To create new layers that are frozen in all viewports

1 Click a layout tab.

2 Click Home tab ➤ Layer panel ➤ Layer Properties.

3 Click the New Layer button to create a layer.

4 Rename the new layer.

5 Click the icon in the Freeze column to change the layer's state to frozen. The sun icon means that a layer is thawed; the snowflake icon means that a layer is frozen.

6 Click OK.

Toolbar: Layers 
Command entry: LAYER, VPLAYER

Quick Reference

Commands

LAYER

Manages layers and layer properties.

VPLAYER

Sets layer visibility within viewports.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Screen Objects in Layout Viewports

Screening refers to applying less ink to an object when it is plotted. The object appears dimmer on the screen and on the plotted paper. Screening can be used to help differentiate objects in a drawing without changing the objects' color properties.

To assign a screening value to an object, you must assign a plot style to the object, and then define the screening value in that plot style.

You can assign a screening value from 0 to 100. The default setting, 100, means no screening is applied, and the object is displayed with normal ink intensity. A screening value of 0 means the object contains no ink and is thus invisible in that viewport.

See also:

- Set Options for Plotted Objects on page 1809

To apply screening to objects in a layout viewport

1. Click File menu ➤ Plot Style Manager.
2. Right-click a CTB or STB file. Click Open.
3. In the Plot Style Table Editor, Form View tab, select a plot style to change.
4. In the Screening box, enter an intensity value between 1 and 100.
5. Click Save & Close.
6. Click Output tab ➤ Plot panel ➤ Page Setup Manager.
7. In the Page Setup dialog box, Plot Device tab, select the edited plot style table from the Plot Style Table (Pen Assignments) list.
8. Double-click within the layout viewport that contains the objects whose screening you want to change.
9. Select the objects whose plot style you want to change.
10. Right-click in the drawing area and click Properties.
11. In the Properties palette, use one of the following methods:
    - If you are using named plot style tables, next to Plot Style, select the plot style you edited in the Plot Style Table Editor. If that plot style is not listed, select Other and set Active Plot Style Table to the plot style you edited in the Plot Style Table Editor. Select the edited plot style from the Plot Styles list in the Select Plot Style dialog box.
    - If you are using color-dependent plot style tables, next to Color, select the color whose plot style you edited in the Plot Style Table Editor.

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PROPERTIES

Controls properties of existing objects.

System Variables

No entries

Utilities

No entries

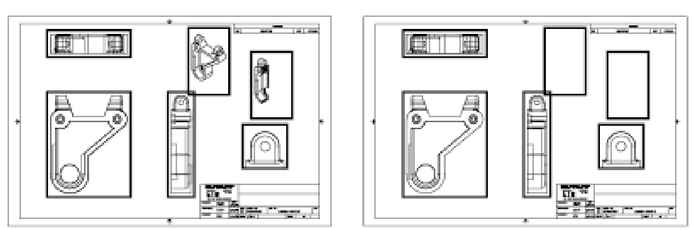
Command Modifiers

No entries

Turn Layout Viewports On or Off

You can save time by turning some layout viewports off or by limiting the number of active viewports.

Displaying a large number of active layout viewports can affect your system's performance as the content of each layout viewport regenerates. You can save time by turning some layout viewports off or by limiting the number of active viewports. The following illustration shows the effects of turning off two layout viewports.



all viewports on   two viewports off

New layout viewports are turned on by default. If you turn off the layout viewports you aren't using, you can copy layout viewports without waiting for each one to regenerate.

If you don't want to plot a layout viewport, you can turn the layout viewport off.

To turn viewports on or off using the Properties palette

1 Make sure you are on a layout tab in paper space.

2 Click the border of the viewport to turn on or off.

3 Click Home tab ➤ Layer panel ➤ Layer Properties.

4 In the Properties palette, under Misc, select On, and then select Yes or No to turn the viewport on or off.

   For a nonrectangular viewport, select All (2) in the Properties palette, and then select Viewport (1) before changing any viewport properties.

Toolbar: Standard 
Command entry: PROPERTIES
Shortcut menu: Select the viewport and right-click in the drawing area. Click Display Viewport Objects.

Quick Reference

Commands

MVIEW

Creates and controls layout viewports.

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PROPERTIES

Controls properties of existing objects.

VPORTS

Creates multiple viewports in model space or paper space.

System Variables

MAXACTVP

Sets the maximum number of viewports that can be active at one time in a layout.

Utilities

No entries

Command Modifiers

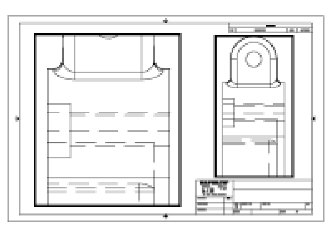
No entries

Scale Linetypes in Layout Viewports

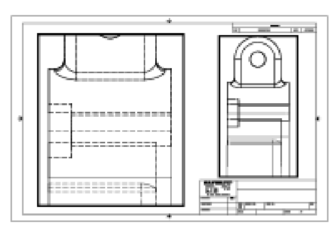
You can scale linetypes in paper space either based on the drawing units of the space in which the object was created or based on the paper space units.

You can set the PSLTSCALE system variable to maintain the same linetype scaling for objects displayed at different zoom factors in a layout and in a layout viewport. For example, with PSLTSCALE set to 1 (default), set the current linetype to dashed, and then draw a line in a paper space layout. In the layout, create a viewport with a zoom factor of 1x, make that layout viewport current, and then draw a line using the same dashed linetype. The dashed lines should appear to be the same. If you change the viewport zoom factor to 2x, the linetype scaling for the dashed line in the layout and the dashed line in the layout viewport will be the same, regardless of the difference in the zoom factor.

With PSLTSCALE turned on, you can still control the dash lengths with LTSCALE and CELTSCALE. In the following illustration, the pattern of the linetypes in the drawing on the left has been scaled to be the same regardless of the scale of the view. In the drawing on the right, the scale of the linetypes matches the scale of each view.



PSLTSCALE=1, dashes scaled to paper space



PSLTSCALE=0, dashes scaled to space where they were created

See also:

- Set the Lineweight Scale for a Layout on page 1763

To scale linetypes globally in paper space

1. Click Home tab ➤ Properties panel ➤ Linetype.
2. In the Linetype Manager, click Show Details.
3. Under Global Scale Factor, enter a global scale to apply to the linetypes.
4. Click OK.

Command entry: LINETYPE

Quick Reference

Commands

LINETYPE

Loads, sets, and modifies linetypes.

System Variables

PSLTSCALE

Controls the linetype scaling of objects displayed in paper space viewports.

Utilities

No entries

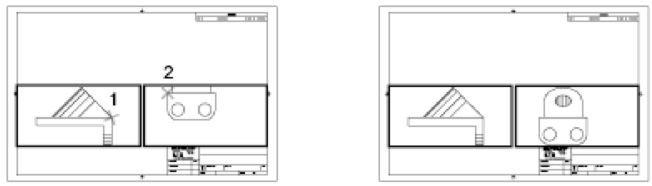
Command Modifiers

No entries

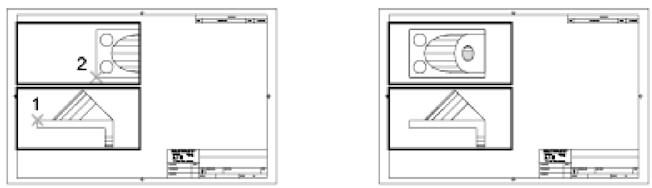
Align Views in Layout Viewports

You can arrange the elements of your drawing by aligning the view in one layout viewport with the view in another viewport.

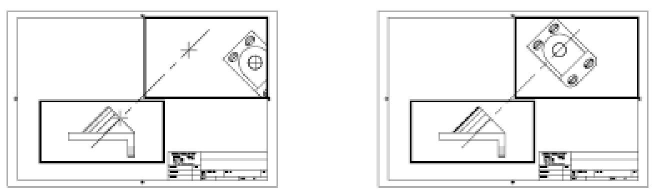
For angled, horizontal, and vertical alignments, you can move each layout viewport relative to distances defined by the model-space geometry displayed.



Horizontal alignments



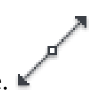
Vertical alignments


Angled alignments

To adjust the views on a layout with precision, you can create construction geometry, use object snaps on the model-space objects displayed in layout viewports, or use one of the cursor constraint features available on the status bar.

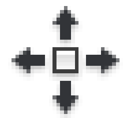
To align objects between viewports using a construction line

1 Make sure you are on a layout tab.

2 Click Home tab ➤ Draw panel ➤ Construction Line. 

3 Specify a point in the first viewport. Specify a second point to determine a line for the alignment.

 Choose a point that can be aligned with objects in the second viewport. Use object snaps for precision.

4 Click Home tab ➤ Modify panel ➤ Move. 

5 Select the viewport to align to the first viewport. Press ENTER.

6 When prompted for a base point, specify a point in the second viewport. Choose a point that corresponds with the point selected in the first viewport.

7 When prompted for the second point, hold down SHIFT and right-click. Click Object Snap ➤ Perpendicular.Click the construction line you created.

   The first and second viewports, and the objects in the viewports, are aligned.

NOTE When aligning objects in viewports, the scale of the viewports should be the same.

To align objects between viewports using MVSETUP

1 At the command prompt, enter **mvsetup**.

2 Enter **a** (Align).

3 Select one of the following alignments:

   - **Horizontal.** Aligns a point in one viewport horizontally with a base point in another viewport.
   - **Vertical.** Aligns a point in one viewport vertically with a base point in another viewport.
   - **Angled.** Aligns a point in one viewport at a specified distance and angle from a base point in another viewport.

4 Make sure the viewport with the view that is to remain stationary is current. Then specify a base point.

5 Select the viewport with the view you want to realign. Then specify an alignment point in that view.

6 For angled alignments only, specify a distance and displacement angle from the base point to the alignment point in the second viewport.

Quick Reference

Commands

MOVE

Moves objects a specified distance in a specified direction.

MVSETUP

Sets up the specifications of a drawing.

UCS

Manages user coordinate systems.

UCSICON

Controls the visibility and placement of the UCS icon.

UCSMAN

Manages defined user coordinate systems.

System Variables

UCSICON

Controls the visibility and placement of the UCS icon.

UCSVP

Determines whether the UCS in viewports remains fixed or changes to reflect the UCS of the current viewport.

Utilities

No entries

Command Modifiers

No entries

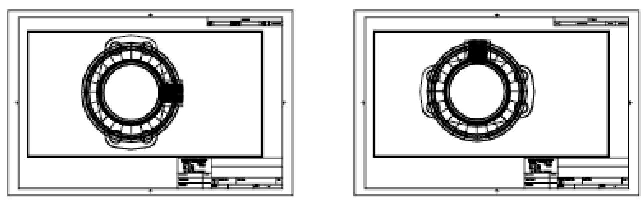
Rotate Views in Layout Viewports

You can rotate an entire view within a layout viewport with the VPROTATEASSOC system variable.

When VPROTATEASSOC is set to 1, the view within a viewport is rotated with the viewport. When VPROTATEASSOC is set to 0, the view remains when the viewport is rotated.

You can also rotate an entire view within a layout viewport by changing the UCS and using the PLAN command.

With the UCS command, you can rotate the *XY* plane at any angle around the Z axis. When you enter the PLAN command, the view rotates to match the orientation of the *XY* plane.



original view rotated view

Another way is touse the Align and then Rotate View options in the MVSETUP command.

NOTE The ROTATE command rotates individual objects only and should not be used to try to rotate a view.

To rotate a view within a viewport

1. At the Command prompt, enter **vprotateassoc**.
2. Set the value to **1**.
3. Make sure you are on a layout tab.
4. Select the viewport with the view you want to rotate.
5. Enter **r** (Rotate) to rotate the view to a specified angle or with two points.
6. Specify a base point for the rotation.
7. Specify the rotation angle or specify a second point to determine the angle of rotation.
   The entire view rotates within the viewport.

NOTE When VPROTATEASSOC is set to 0, the view within the viewport is not rotated when the viewport is rotated.

To rotate a view by changing the UCS

1 Make sure you are on a layout tab.

2 Double-click within the viewport whose objects you want to rotate.

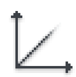
3 Make sure that the current UCS is parallel to the plane of rotation (the UCS icon should look normal). If the UCS is not parallel to the plane of rotation, Click View tab ➤ Coordinates panel ➤ View..

4 Click View tab ➤ Coordinates panel ➤ View.

   To rotate the view 90 degrees clockwise, enter **90**. To rotate the view 90 degrees counter-clockwise, enter **-90**.

5 Click View menu ➤ 3D Views ➤ Plan View ➤ Current UCS.

   The entire view rotates within the viewport. You may need to specify the scale of the viewport again.

Toolbar: UCS 
Command entry: UCS

To rotate a layout view using MVSETUP

1 On a layout, enter **mvsetup** at the command prompt.

2 Enter **a** (Align).

3 Enter **r** (Rotate) to rotate the view to a specified angle or with two points.

4 If multiple viewports are available on the layout, click the viewport with the view that you want to rotate.

5 Specify a base point for the rotation.

6 Specify the rotation angle or specify a second point to determine the angle of rotation.

   The entire view rotates within the viewport.

7 To restore the previous UCS, enter **ucs** and then **p** (Previous).

Quick Reference

Commands

MVSETUP

Sets up the specifications of a drawing.

PLAN

Displays the plan view of a specified user coordinate system

UCS

Manages user coordinate systems.

UCSICON

Controls the visibility and placement of the UCS icon.

UCSMAN

Manages defined user coordinate systems.

System Variables

UCSICON

Controls the visibility and placement of the UCS icon.

UCSVP

Determines whether the UCS in viewports remains fixed or changes to reflect the UCS of the current viewport.

VIEWTWIST

Stores the view rotation angle for the current viewport measured relative to the WCS.

VPROTATEASSOC

Controls whether the view within a viewport is rotated with the viewport when the viewport is rotated.

Utilities

No entries

Command Modifiers

No entries

Reuse Layouts and Layout Settings

When you create a layout, you can choose to apply the information from an existing template.

A layout template is a layout imported from a DWG or DWT file. When you create a layout, you can choose to apply the information from an existing template. The program has sample layout templates to use when you design a new layout environment. The paper space objects and page setup in the existing template are used in the new layout. Thus, the layout objects, including any viewport objects, are displayed in paper space. You can keep any of the existing objects from the template you import, or you can delete the objects. No model space objects are imported.

The layout templates are identified with a *.dwt* file extension. However, a layout template or layout from any drawing or drawing template can be imported into the current drawing.

Save a Layout Template

Any drawing can be saved as a drawing template (DWT file), including all of the objects and layout settings. You can save a layout to a new DWT file by choosing the Save As option of the LAYOUT command. The template file is saved in the drawing template file folder as defined in the Options dialog box, Support tab. The layout template has a *.dwt* or *.dwg* extension like a drawing template or drawing file, but it contains little information not essential to the layout.

When you create a new layout template, any named items, such as blocks, layers, and dimension styles, that are used in the layout are saved with the template. These definition table items are imported as part of the layout settings if you import this template into a new layout. It is recommended that you use the Save As option of the LAYOUT command to create a new layout template. When you use the Save As option, unused definition table items are not saved with the file; they are not added to the new layout into which you import the template.

If you insert a layout from a drawing or template that was not created using the Save As option of the LAYOUT command, definition table items that are used in the drawing but not in the layout are inserted with the layout. To eliminate unnecessary definition table items, use the PURGE command.

Insert a Layout Using DesignCenter

Using DesignCenter™, you can drag a layout with its objects from any drawing into the current drawing.

When you use DesignCenter to insert a layout into a drawing, a new layout is created that includes all of the paper space objects, definition tables, and block definitions from the source layout. You can delete unneeded paper space objects. To eliminate any unnecessary definition table information from the new layout, use the PURGE command.

To create a layout using a layout template

1. Click Insert menu ➤ Layout ➤ Layout From Template.
2. In the Select Template From File dialog box, select a drawing template file from the list.
3. Click Open.
4. In the Insert Layout(s) dialog box, select the layout template from the list. Click OK.

   A new layout is created using the layout template you selected. The new layout is assigned the name Layout with the next number in the sequence and with the name of the imported layout attached.

   For example, if you insert a layout called ANSI D from a layout template and you already have two layouts in your drawing called Layout1 and Layout2, the new layout is called Layout3 - ANSI D.

Toolbar: Layouts 
Command entry: LAYOUT
Shortcut menu: Right-click the layout tab. Click From Template.

To save a layout template

1. At the command prompt, enter **layout**.
2. At the prompt, enter **sa** to save the current layout as a template.

3 Enter the name of the layout you are saving.

4 In the Create Drawing File dialog box, enter a name for the drawing template file you are saving.

5 In Files of Type, select Drawing Template File (*.dwt).

6 Click Save.

To insert a layout using DesignCenter

1 Click View tab ➤ Palettes panel ➤ DesignCenter.

2 In the tree view, find the drawing that contains the layout you want to reuse.

3 Double-click the drawing name to expand the options beneath it.

4 Select the Layouts icon to display the individual layouts in the content area.

5 Use one of the following methods to insert the layout into the current drawing:
   - Drag the layout icon from the content area into the drawing.
   - Select a layout in the content area and right-click. Click Add Layout(s).
   - Double-click the layout in the content area.

Toolbar: Standard 
Command entry: ADCENTER

Quick Reference

Commands

ADCENTER

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

LAYOUT

Creates and modifies drawing layout tabs.

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PCINWIZARD

Displays a wizard to import PCP and PC2 configuration file plot settings into the Model tab or current layout.

PURGE

Removes unused items, such as block definitions and layers, from the drawing.

System Variables

TDCREATE

Stores the local time and date the drawing was created.

TDUCREATE

Stores the universal time and date that the drawing was created.

TDUPDATE

Stores the local time and date of the last update/save.

TDUUPDATE

Stores the universal time and date of the last update or save.

Utilities

No entries

Command Modifiers

No entries

Work with Sheets in a Sheet Set

14

With the Sheet Set Manager, you can organize drawing layouts into named sheet sets. The sheets in a sheet set can be transmitted, published, and archived as a unit.

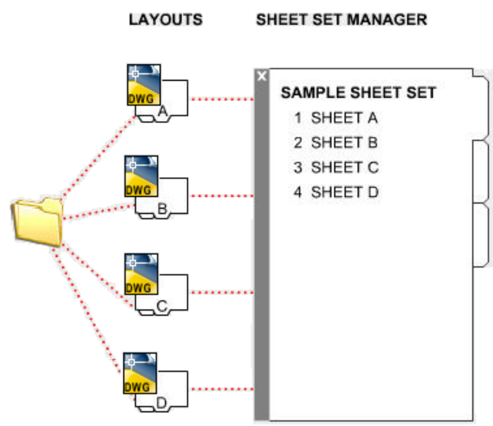
Sheet sets facilitate the organization and management of drawings in a project, and improve the communication in a work group.

Quick Start for Sheet Sets

A sheet set is an organized collection of *sheets* from several drawing files. A sheet is a selected layout from a drawing file.

Sets of drawings are the primary deliverable for most design groups. Sets of drawings communicate the overall design intent of a project and provide the documentation and specifications for the project. However, managing sets of drawings manually can be complicated and time consuming.

With the Sheet Set Manager, you can manage drawings as *sheet sets*. A sheet set is an organized and named collection of sheets from several drawing files. A sheet is a selected layout from a drawing file. You can import a layout from any drawing into a sheet set as a numbered sheet.



You can manage, transmit, publish, and archive sheet sets as a unit.

Quick Reference

Commands

NEWSHEETSET

Creates a new sheet set data file that manages drawing layouts, file paths, and project data.

OPENSHEETSET

Opens a selected sheet set.

SHEETSET

Opens the Sheet Set Manager.

SHEETSETHIDE

Closes the Sheet Set Manager.

System Variables

SSFOUND

Displays the sheet set path and file name if a search for a sheet set is successful.

SSLOCATE

Controls whether the sheet set associated with a drawing is located and opened when the drawing is opened.

SSMAUTOOPEN

Controls the display behavior of the Sheet Set Manager when a drawing associated with a sheet is opened.

SSMPOLLTIME

Controls whether plotting a sheet set, multi-sheet plot file, or plot spool file can be interrupted by other plot jobs.

SSMSHEETSTATUS

Controls how the status data in a sheet set is refreshed.

SSMSTATE

Indicates whether the Sheet Set Manager window is open or closed.

Utilities

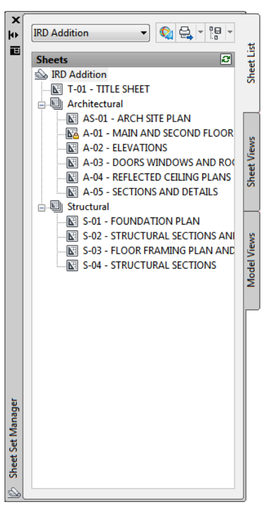
No entries

Command Modifiers

No entries

Understand the Sheet Set Manager Interface

Using the controls in the Sheet Set Manager, you can create, organize, and manage sheets in a sheet set.



You can use the following tabs and controls in the Sheet Set Manager:

Sheet Set Control. Lists menu options to create a new sheet set, open an existing sheet set, or switch between open sheet sets.

Sheet List Tab. Displays an organized list of all sheets in the sheet set. Each sheet in a sheet set is a specified layout in a drawing file.

Sheet Views Tab. Displays an organized list of all sheet views in the sheet set. Only sheet views created with AutoCAD 2005 and later are listed.

Model Views Tab. Lists the paths and folder names for the drawings containing model space views to be used in the sheet set.

- Click a folder to list the drawing files that are located in that folder.
- Click a drawing file to list the named model space views that are available for placement in the current sheet.
- Double-click a view to open the drawing containing the view.
- Right-click or drag a view to place it in the current sheet.

Buttons. Provides convenient access for the most commonly used operations for the currently selected tab.

Tree View. Displays the contents of a tab.

Details or Preview. Displays either descriptive information or a thumbnail preview of the currently selected item in the tree view.

Actions Used in the Tree View

You can use the following actions in the tree view:

- Right-click to access shortcut menus of the operations that are relevant to the currently selected item.
- Double-click items to open them. This is a convenient method for opening drawing files from the Sheet List tab or from the Model Views tab. You also double-click items in the tree view to expand or collapse them.
- Click one or more items to select them for operations such as opening, publishing, or transmitting.
- Click a single item to display descriptive information or a thumbnail preview of a selected sheet, view, or drawing file.
- Drag items within the tree view to reorder them.

NOTE To use the Sheet Set Manager effectively, right-click items in the tree view to access relevant shortcut menus. For access to shortcut menus in the drawing area that are needed for sheet set operations, the Shortcut Menus in Drawing Area must be checked in the Options dialog box, User Preferences tab.

Quick Reference

Commands

NEWSHEETSET

Creates a new sheet set data file that manages drawing layouts, file paths, and project data.

OPENSHEETSET

Opens a selected sheet set.

SHEETSET

Opens the Sheet Set Manager.

SHEETSETHIDE

Closes the Sheet Set Manager.

System Variables

SSFOUND

Displays the sheet set path and file name if a search for a sheet set is successful.

SSLOCATE

Controls whether the sheet set associated with a drawing is located and opened when the drawing is opened.

SSMAUTOOPEN

Controls the display behavior of the Sheet Set Manager when a drawing associated with a sheet is opened.

SSMPOLLTIME

Controls whether plotting a sheet set, multi-sheet plot file, or plot spool file can be interrupted by other plot jobs.

SSMSHEETSTATUS

Controls how the status data in a sheet set is refreshed.

SSMSTATE

Indicates whether the Sheet Set Manager window is open or closed.

Utilities

No entries

Command Modifiers

No entries

Create and Manage a Sheet Set

There are several methods for setting up and organizing a sheet set. You can also include relevant information with a sheet set and its components.

Create a Sheet Set

You create a sheet set with the Create a Sheet Set wizard. In the wizard, you can either create a sheet set from scratch based on existing drawings or use an example sheet set as a template.

Layouts from specified drawing files are imported into the sheet set. The associations and information that define a sheet set are stored in a sheet set data (DST) file.

When creating a new sheet set using the Create Sheet Set wizard, a new folder is created as the default sheet set storage location. This new folder, named *AutoCAD Sheet Sets*, is located in the *My Documents* folder. The default location for the sheet set file can be changed, and it is recommended that the DST file is stored with the project files.

NOTE The DST file should be stored in a network location that is accessible to all sheet set users on the network and mapped using the same logical drive. It is strongly recommended that you store the DST and the sheet drawings in the same folder. If an entire sheet set needs to be moved, or a server or folder name changes, the DST file will still be able to locate the sheets using relative path information.

Preparation Tasks

Before you begin creating a sheet set, you should complete the following tasks:

- **Consolidate drawing files.** Move the drawing files to be used in the sheet set into a small number of folders. This will simplify sheet set administration.
- **Eliminate multiple layout tabs.** Each drawing you plan to use in the sheet set should have only one layout to be used as a sheet in the sheet set. This is important for access to sheets by multiple users. Only one sheet in each drawing can be open at a time.
- **Create a sheet creation template.** Create or identify a drawing template (DWT) file to be used by the sheet set for creating new sheets. This drawing template file is called the *sheet creation template*. You specify this template file in the Sheet Set Properties dialog box or the Subset Properties dialog box.
- **Create a page setup overrides file.** Create or identify a DWT file to store page setups for plotting and publishing. This file, called the *page setup overrides file*, can be used to apply a single page setup to all sheets in a sheet set, overriding the individual page setups stored in each drawing.

NOTE Although it is possible to use several layouts from the same drawing file as separate sheets in a sheet set, it is not recommended. This makes concurrent access to each layout by multiple users impossible. This practice can also reduce your management options and can complicate the organization of your sheet sets.

Create a Sheet Set from an Example Sheet Set

In the Create Sheet Set wizard, when you choose to create a sheet set from an example, the example sheet set provides the organizational structure and default settings for the new sheet set. You can also specify that folders are created corresponding to the subset storage paths of the sheet set.

After you create an empty sheet set with this option, you can import layouts or create sheets individually.

Create a Sheet Set from Existing Drawing Files

In the Create Sheet Set wizard, when you choose to create a sheet set from existing drawing files, you specify one or more folders that contain drawing files. With this option, you can specify that the subset organization for the sheet set duplicates the folder structure of the drawing files. The layouts from these drawings can be imported into the sheet set automatically.

You can easily add more folders containing drawings by clicking the Browse button for each additional folder.

Backup and Recover Sheet Set Data Files

The data stored in the sheet set data file represents a significant amount of work, so you should take the same care to create backups of DST files as you do for drawing files.

In the unlikely event of DST file corruption or a major user error, the previously saved sheet set data file can be recovered. Every time the sheet set data file is opened, the current sheet set data file is copied to a backup file (DS$). This backup file has the same file name and is located in the same folder as the current sheet set data file.

To recover the previous version of the sheet set data file, first make sure that no one else on your network is working on the sheet set. Then, it is recommended that you copy the existing DST file to another file name. Finally, rename the backup file from the DS$ file extension to the DST file extension.

To open the Sheet Set Manager

- Click View tab ➤ Palettes panel ➤ Sheet Set Manager. 

Toolbar: Standard 
Command entry: SHEETSET

To create a new sheet set

1 To create a new sheet set, do *one* of the following:

  - Click View tab ➤ Palettes panel ➤ Sheet Set Manager. 
  - In the Sheet Set Manager, click the Sheet Set control. Click New Sheet Set.

2 Follow the steps in the Create Sheet Set wizard.

Command entry: NEWSHEETSET

To open a sheet set

1 To open a sheet set, do *one* of the following:

- Click View tab ➤ Palettes panel ➤ Sheet Set Manager.
- In the Sheet Set Manager, click the Sheet Set control. Click Open.
- Double-click a sheet set data (DST) file.

2 In the Open Sheet Set dialog box, navigate to a folder that contains a DST file. Click the DST file and then click open.

The Sheet Set Manager displays the sheet set data.

NOTE You can open several sheet sets and then use the Sheet Set control to switch between them.

Toolbar: Standard
Command entry: SHEETSET

To close a sheet set

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager.

2 In the Sheet Set Manager, Sheet List tab, right-click the sheet set node (at the top of the list). Click Close Sheet Set.

The sheet set is no longer displayed in the Sheet Set Manager.

NOTE Close a sheet set to reduce the number of sheet sets listed in the Sheet Set control. However, if you close a sheet set while you still have sheets in the sheet set open, you will not be able to update the Sheet List table and certain fields.

Toolbar: Standard
Command entry: SHEETSET

Quick Reference

Commands

NEWSHEETSET

Creates a new sheet set data file that manages drawing layouts, file paths, and project data.

OPENSHEETSET

Opens a selected sheet set.

SHEETSET

Opens the Sheet Set Manager.

SHEETSETHIDE

Closes the Sheet Set Manager.

System Variables

SSFOUND

Displays the sheet set path and file name if a search for a sheet set is successful.

SSLOCATE

Controls whether the sheet set associated with a drawing is located and opened when the drawing is opened.

SSMAUTOOPEN

Controls the display behavior of the Sheet Set Manager when a drawing associated with a sheet is opened.

SSMPOLLTIME

Controls whether plotting a sheet set, multi-sheet plot file, or plot spool file can be interrupted by other plot jobs.

SSMSHEETSTATUS

Controls how the status data in a sheet set is refreshed.

SSMSTATE

Indicates whether the Sheet Set Manager window is open or closed.

Utilities

No entries

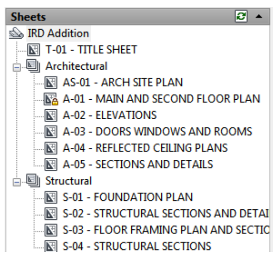
Command Modifiers

No entries

Organize a Sheet Set

With a large sheet set, you will find it essential to organize sheets and views in the tree view.

On the Sheet List tab, sheets can be arranged into collections called *subsets*. On the Sheet Views tab, views can be arranged into collections called *categories*.



Use Sheet Subsets

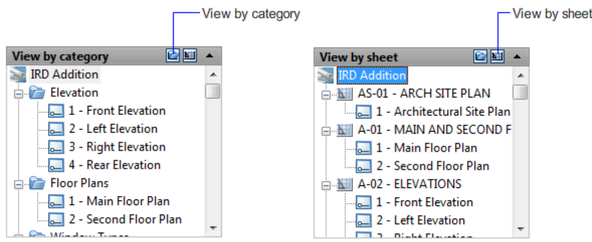
Sheet subsets are often associated with a discipline such as architecture or mechanical design. For example, in architecture, you might use a subset named Structural; and in mechanical design, you might use a subset called Standard Fasteners. In some cases, you might also find it useful to create subsets associated with a review or completion status.

You can nest subsets into other subsets as needed. After you create or import sheets or subsets, you can reorder them by dragging them in the tree view.

Use View Categories

View categories are often associated with a function. For example, in architecture, you might use a view category called Elevations; and in mechanical design, you might use a view category called Exploded.

You can display views either by category or by the sheet on which they are located.



You can nest categories into other categories as needed. To move a view to a different category, drag them in the tree view or use the Set Category shortcut menu option.

To reorder sheets in the sheet list

- Click View tab ➤ Palettes panel ➤ Sheet Set Manager. 
- In the Sheet Set Manager, on the Sheet List tab, drag a sheet up or down the list.

The sheet is relocated above or below its previous location on the sheet list. The sheet may also be moved into or out of a sheet subset.

Toolbar: Standard 
Command entry: SHEETSET

To create a new subset in the sheet list

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager. 

2 In the Sheet Set Manager, Sheet List tab, right-click the sheet set node (at the top of the list), or an existing subset. Click New Subset.

3 In the Subset Properties dialog box, under Subset Name, enter the name of the new subset. Click OK.

You can drag the new subset anywhere on the sheet list, even under other subsets.

NOTE If you want to create a subset under an existing subset, you can right-click the existing subset. On the shortcut menu, click New Subset.

Toolbar: Standard
Command entry: SHEETSET

To create a new view category in the view list

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager.

2 In the Sheet Set Manager, Sheet Views tab, click the View by Category button.

3 Right-click the sheet set node (at the top of the list). Click New View Category.

4 In the View Category dialog box, under Category Name, enter the name of the new view category.

5 If a list of blocks is displayed, select the callout blocks to be used for the views in this view category. You can add blocks to the list by clicking the Add Blocks button.

Toolbar: Standard
Command entry: SHEETSET

To remove a subset from the sheet list

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager.

2 In the Sheet Set Manager, Sheet List tab, drag all sheets out of the subset that you want to remove.

3 Right-click the subset that you want to remove. Click Remove Subset.

Toolbar: Standard
Command entry: SHEETSET

To remove a view category from the view list

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager. 

2 In the Sheet Set Manager, Sheet Views tab, click the View by Category button.

3 Drag all views out of the view category that you want to remove.

4 Right-click the view category that you want to remove. Click Remove Category.

Toolbar: Standard 
Command entry: SHEETSET

Quick Reference

Commands

NEWSHEETSET

Creates a new sheet set data file that manages drawing layouts, file paths, and project data.

OPENSHEETSET

Opens a selected sheet set.

SHEETSET

Opens the Sheet Set Manager.

SHEETSETHIDE

Closes the Sheet Set Manager.

System Variables

SSFOUND

Displays the sheet set path and file name if a search for a sheet set is successful.

SSLOCATE

Controls whether the sheet set associated with a drawing is located and opened when the drawing is opened.

SSMAUTOOPEN

Controls the display behavior of the Sheet Set Manager when a drawing associated with a sheet is opened.

SSMPOLLTIME

Controls whether plotting a sheet set, multi-sheet plot file, or plot spool file can be interrupted by other plot jobs.

SSMSHEETSTATUS

Controls how the status data in a sheet set is refreshed.

SSMSTATE

Indicates whether the Sheet Set Manager window is open or closed.

Utilities

No entries

Command Modifiers

No entries

Create and Modify Sheets

There are several options in the Sheet Set Manager for creating sheets and adding views either through a shortcut menu or one of the tab buttons. Modifying a sheet should always be done from an open sheet set.

Following are descriptions of common sheet operations. You can access commands by right-clicking an item in the tree view to display the relevant shortcut menu.

- **Import layout as sheet.** After you create a sheet set, you can import one or more layouts from existing drawings. You can initialize a layout by clicking on its tab to activate the previously unused layout. A layout does not contain any plot settings before initialization. Once initialized, layouts can be drawn upon, published, and added to sheet sets as sheets (after the drawing has been saved). This is a fast method for creating multiple sheets from layouts in several drawings. In the current drawing, you can drag a layout tab directly onto the Sheets area of the Sheet List tab in the Sheet Set Manager.

- **Create a new sheet.** As an alternative to importing existing layouts, you can create a new sheet. When you place views in this sheet, the drawing files associated with the views are attached as xrefs to the sheet drawing. The sheet drawing file is created using either the AutoCAD 2004 format or the AutoCAD 2007 format, depending on the format specified on the Open and Save tab in the Options dialog box.

- **Modify a sheet.** Double-click a sheet on the Sheet List tab to open a drawing from the sheet set. Use SHIFT or CTRL to select multiple sheets. To review a sheet, use the shortcut menu to open a drawing in read-only mode.

  NOTE Modifying a sheet should always be done using an open sheet set in the Sheet Set Manager. This ensures that all data associated with the sheet are updated.

- **Rename and renumber a sheet.** After you create a sheet, you can change the sheet title and the sheet number. You can also specify a different drawing file associated with the sheet.

  NOTE If you change the layout name, the corresponding sheet title in the sheet set is also updated or vice versa.

- **Remove a sheet from a sheet set.** Removing a sheet from a sheet set disassociates the sheet from the sheet set, but does not delete the drawing file or the layout.

- **Reassociate a sheet.** If you move a sheet to a different folder, you should reassociate the sheet to the sheet set with the Sheet Properties dialog box to correct the path. For any relocated sheet drawing, the paths for Expected Layout and Found Layout are displayed in the Sheet Properties dialog box. To reassociate the sheet, click the path in Expected Layout and then click to navigate to the new location of the sheet.

NOTE You can quickly confirm whether a sheet is in the expected folder by looking at Details at the bottom of the Sheet List tab. If the selected sheet is not in the expected location, path information for both Expected Location and Found Location is displayed in Details.

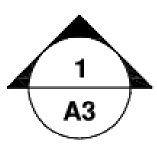
- **Add a view to a sheet.** From the Model Views tab, you can easily add a view to a sheet by placing a named model space view or the entire drawing onto the current sheet.

NOTE After creating a named model space view, you must save the drawing to add the view to the Model Views tab. Click Refresh on the Model Views tab to update the Sheet Set Manager tree view.

- **Add label blocks to views.** With the Sheet Set Manager, you can label views and details automatically as you place them. Labels contain data associated with the referenced view.

- **Add callout blocks to views.** *Callout blocks* is the term for the symbols that reference other sheets. Callout blocks have many industry-specific names such as reference tags, detail keys, detail makers, building section keys, and so on. Callout blocks contain data associated with the sheet and view that are referenced.

NOTE When you place a callout block with fields or a view on a sheet, make sure that the current layer is unlocked.



- **Create a title sheet and table of contents.** The first sheet in a sheet set will usually be a title sheet that includes a description of the sheet set and a table that lists all the sheets in the sheet set. You can create this table, called a *sheet list table*, on an open sheet. The table automatically includes

all the sheets in the sheet set. Once a sheet list table is created, you also have options to edit, update, or delete the cell content of the table.
You can create a sheet list table from the shortcut menu of sheet set subsets and sheets only when a sheet is open. You can also create sheet list tables for multiple subsets and sheets. Any sheets added to a subset later on are automatically added to the sheet list table.

NOTE For access to shortcut menus in the drawing area that are needed for sheet list table operations, the Shortcut Menus in Drawing Area must be checked in the Options dialog box, User Preferences tab.

Create Callout Blocks and Label Blocks (Advanced)

If you create a block to be used as a callout block or label block in a sheet set, you can use a placeholder field to display information such as view title or sheet number. The callout or label block must be defined in a DWG or DWT file that is specified in the Sheet Set Properties dialog box. Later, you can insert the callout or label block from a shortcut menu on the Sheet Views tab in the Sheet Set Manager.

For the field to display the correct information about a view or sheets on which you later insert it, the field must be included within a block attribute, not text, when you define the block. To create the block attribute definition, insert a placeholder field as the value, select the Preset option, and specify a tag.

NOTE If you create your own label blocks and callout blocks, set any attribute definitions to Preset to avoid prompts when placing these blocks in a drawing.

For more information about fields, see Insert Fields on page 1532.

Place a Sheet View (Advanced)

The Sheet Set Manager automates and enhances the process for adding views to a sheet. A view on a sheet, called a *sheet view*, consists of several coincident entities: an xref or geometry in model space, a layout viewport on a sheet, and a named view in paper space.

- The sheet view can display model space from a different drawing file. In this case, that drawing is attached as an xref in your current drawing. The layers of that drawing file are displayed only in the sheet view that you create.

NOTE The xref is attached using a relative path. If you need to change the path to a fully specified (absolute) path, use the External References palette.

- A layout viewport that displays the model space view is created on your current sheet.
- A named view that encompasses the area of the layout viewport is created in paper space.

When you place a sheet view on a sheet, all the layers in the current drawing (including layer 0) are frozen in the new viewport created by the view. The layers are shown as frozen in the VP Freeze column of the Layer Properties Manager.

If you need to remove a sheet view from a sheet, you can delete the layout viewport to remove the view. However, to remove all unused items, you need to detach the xref and delete the named paper space view.

NOTE The easiest method for removing a sheet view immediately after placing it is to use UNDO.

See also:

- Create Multiple-View Drawing Layouts (Paper Space) on page 431

To import a layout from the current drawing as a sheet

1. If the Model and layout tabs are not visible at the bottom of the drawing area, right-click the Model button on the status bar and click Display Layout and Model tabs.
2. Do *one* of the following:
   - Right-click a layout tab and click Import Layout as Sheet.
   - Click View tab ➤ Palettes panel ➤ Sheet Set Manager.
   - Drag a layout tab onto the Sheet Set Manager, Sheet List Tab, Sheets area.
3. Click Import Checked.

Toolbar: Standard 

Command entry: SHEETSET

To import layouts from several drawings as sheets

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager.

2 In the Sheet Set Manager, Sheet List tab, right-click the sheet set node, a subset node, or a sheet node. Click Import Layout as Sheet.

3 In the Import Layouts as Sheets dialog box, click Browse for Drawings and navigate to the drawing that you want to use.

  To select several drawings, use SHIFT or CTRL when you click on the drawing files.

4 Click the check boxes of the layouts to be imported as sheets in the current sheet set. Click Import Checked.

5 (Optional) On the Sheet List tab, right-click a newly imported sheet. Click Rename & Renumber.

6 (Optional) In the Rename & Renumber Sheet dialog box, make any changes to the sheet number and sheet title. Click OK.

  NOTE Changes to the sheet number and sheet title have no effect on the drawing file name unless a file rename option is checked.

Toolbar: Standard
Command entry: SHEETSET

To create a new sheet

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager.

2 In the Sheet Set Manager, Sheet List tab, right-click either the sheet set node, a subset node, or a sheet node. Click New Sheet.

3 In the New Sheet dialog box, enter the sheet number and sheet title. Click OK.

  The new sheet is created from the drawing template file specified in Sheet Set Properties for the default new sheet creation template.

NOTE You can change the default folder location for new sheets from the Sheet Set Properties or the Subset Properties dialog box.

Toolbar: Standard
Command entry: SHEETSET

To open a drawing file within a sheet set

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager.
2 In the Sheet Set Manager, open a sheet set.
3 On the Sheet List tab, double-click a sheet. To select several drawings, use SHIFT or CTRL when you click on the drawing files.

   The drawing file for the sheet is opened. Any changes made to the drawing that are relevant to the sheet list are updated in the sheet set data (DST) file.

Toolbar: Standard
Command entry: SHEETSET

To open a drawing file read-only within a sheet set

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager.
2 In the Sheet Set Manager, open a sheet set.
3 On the Sheet List tab, right-click a sheet. Click Open Read-Only.

Toolbar: Standard
Command entry: SHEETSET

To remove a sheet from a sheet set

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager.

2 In the Sheet Set Manager, open a sheet set.

3 On the Sheet List tab, right-click the sheet that you want to remove. Click Remove Sheet.

  The specified sheet is removed from the sheet list. The sheet and the drawing file are not deleted, however. The sheet can be added to a different sheet set.

  NOTE The DWG file associated with a sheet does not have to be accessible for the sheet to be removed from the sheet set.

Toolbar: Standard
Command entry: SHEETSET

To reassociate a sheet in a sheet set

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager.

2 In the Sheet Set Manager, open a sheet set.

3 On the Sheet List tab, open the sheet that you want to reassociate.

4 In the Sheet Set Manager, right-click the sheet. Click Remove Sheet.

5 Save the drawing.

6 In the Sheet Set Manager, right-click the Sheet Set. Click Import Layout as Sheet.

7 In the Import Layout as Sheet dialog box, click Browse for Drawings. Navigate to the drawing that you want to use.

8 Click the check box of the layout to be reassociated as a sheet in the current sheet set. Click Import Checked.

Toolbar: Standard
Command entry: SHEETSET

To add a view to a sheet

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager.

2 In the Sheet Set Manager, open a sheet set.

3 On the Sheet List tab, do *one* of the following:
   - Double-click a sheet to open it.
   - Create a new sheet and open it.

4 On the Model Views tab, click the plus sign (+) next to a folder to list the drawings in the folder.

5 From the list of drawing files, do *one* of the following:
   - To add a model space view to a sheet, click the plus sign (+) next to a drawing file to list its named model space views. Right-click a model space view.
   - To add an entire drawing as a view in a sheet, right-click a drawing file.

6 Click Place on Sheet.

NOTE As an alternative to the previous two steps, you can drag a model space view or a drawing from the Model Views tab to a sheet.

7 Right-click on the sheet. Click the scale for the sheet view.

8 Specify the insertion point for the sheet view.

The specified view is added to the sheet. If a view label block is defined in the sheet set properties, a view label that displays view-specific information is automatically placed on the sheet.

Toolbar: Standard
Command entry: SHEETSET

To add a sheet list table

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager.

2 In the Sheet Set Manager, open a sheet set.

3 Right-click a sheet set name, subset, or multiple sheet set names and subsets. Click Insert Sheet List Table.

4 In the Insert Sheet List Table dialog box, do the following:

- Set the Table Style in the Table Style Settings group.
- On the Table Data tab, specify Title Text for the table and add, remove, or change the order of the column entries.
- On the Subsets and Sheets tab, select the subsets and sheets to be included in the sheet list table.

  NOTE If you add a sheet to a subset later on, you will be automatically prompted to update the sheet list table.

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5 Click OK.

Toolbar: Standard
Command entry: SHEETSET

To edit a sheet list table

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager.

2 In the Sheet List tab, select a cell in an existing sheet list table.

3 Right-click the cell. Click Edit Sheet List Table Settings.

4 In the Edit Sheet List Table Settings dialog box, do one or more of the following:

- Change the Table Style if you don’t like the current format.
- Edit the Title Text for the table.
- Add, remove, or change the order of the column entries.

- Change the data type or heading text for column entries.

5 Click OK to close the dialog box and change the table to the drawing.

Shortcut menu: Select any cell in the sheet list table ➤ Right-click to open the shortcut menu ➤ Edit Sheet List Table Settings.

To update a sheet list table

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager. 

2 In the Sheet List tab, select a cell in an existing sheet list table.

3 Right-click the cell. Click Update Sheet List Table.

Shortcut menu: Select any cell in the sheet list table ➤ Right-click to open the shortcut menu ➤ Update Sheet List Table.

To add a block to be used for sheet view callouts

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager. 

2 In the Sheet Set Manager, open a sheet set.

3 On the Sheet Views tab, right-click the sheet set node.

4 In the Sheet Set Properties dialog box, click Callout Blocks. Click the [...] button.

5 In the List of Blocks dialog box, do *one* of the following:
   - Click a block from the list of blocks.
   - Click the Add button and specify a new block to be added to the list.

6 Click OK to close each dialog box.

Toolbar: Standard 
Command entry: SHEETSET

To change the default block used for sheet view labels

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager.

2 In the Sheet Set Manager, open a sheet set.

3 On the Sheet Views tab, right-click the sheet set node.

4 In the Sheet Set Properties dialog box, click Label Block for Views. Click the [...] button.

5 In the Select Block dialog box, specify a new block to be used as the default view label block.

6 Click OK to close each dialog box.

Toolbar: Standard
Command entry: SHEETSET

To add a callout block to a sheet

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager.

2 In the Sheet Set Manager, open a sheet set.

3 On the Sheet Views tab, right-click the view with which you want to associate a callout. Click Place Callout Block.

4 Specify the insertion point of the callout block.

   The callout block is placed on the sheet. The callout block automatically displays view-specific information about the view with which it is associated.

Toolbar: Standard
Command entry: SHEETSET

To create a sheet list table on a title sheet

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager. 
2 In the Sheet Set Manager, open a sheet set.
3 On the Sheet List tab, double-click the sheet to be used as the title sheet.
4 Right-click the sheet set node. Click Insert Sheet List Table.
5 In the Sheet List Table dialog box, enter the title of the table. Make any formatting changes needed.
6 Click OK.
7 Specify the insertion point of the table.
   The sheet list table automatically generates a list of all the sheets in the sheet set.

Toolbar: Standard 
Command entry: SHEETSET

Quick Reference

Commands

NEWSHEETSET

Creates a new sheet set data file that manages drawing layouts, file paths, and project data.

OPENSHEETSET

Opens a selected sheet set.

SHEETSET

Opens the Sheet Set Manager.

SHEETSETHIDE

Closes the Sheet Set Manager.

System Variables

SSFOUND

Displays the sheet set path and file name if a search for a sheet set is successful.

SSLOCATE

Controls whether the sheet set associated with a drawing is located and opened when the drawing is opened.

SSMAUTOOPEN

Controls the display behavior of the Sheet Set Manager when a drawing associated with a sheet is opened.

SSMPOLLTIME

Controls whether plotting a sheet set, multi-sheet plot file, or plot spool file can be interrupted by other plot jobs.

SSMSHEETSTATUS

Controls how the status data in a sheet set is refreshed.

SSMSTATE

Indicates whether the Sheet Set Manager window is open or closed.

Utilities

No entries

Command Modifiers

No entries

Include Information with Sheets and Sheet Sets

Sheet sets, subsets, and sheets include several types of information. This information, called properties, includes titles, descriptions, file paths, and custom properties that you define.

Different Properties for Different Levels (Owners)

Sheet sets, subsets, and sheets represent different levels of organization, and each of these includes different types of properties. You specify the values for these properties when you create the sheet set, subset, or sheet.

In addition, you can define custom properties for a sheet and a sheet set. The values for custom properties for sheets are typically specific to each sheet. For example, a custom property for a sheet might include the name of the designer. The values for custom properties for a sheet set are typically specific to a project. For example, a custom property for a sheet set might include the contract number.

You cannot create custom properties for subsets.

View and Edit Properties

You can view and edit properties from the Sheet List tab by right-clicking the name of the sheet set, subset, or sheet. On the shortcut menu, click Properties. The properties and values that are displayed in the Properties dialog box depend on what you selected. You can edit the values of the properties by clicking a value.

To edit the properties of a sheet set, subset, sheet, or view category

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager.

2 In the Sheet Set Manager, open a sheet set.

3 Do *one* of the following:

- On the Sheet List tab, right-click the sheet set node, a subset, or a sheet.
- On the Sheet Views tab, right-click a view category.

4 Click Properties.

5 In the dialog box that is displayed, click any field and enter the changes.

6 Click OK.

Toolbar: Standard
Command entry: SHEETSET

To create a custom property for a sheet set

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager.

2 In the Sheet Set Manager, open a sheet set.

3 On the Sheet List tab, right-click the sheet set node. Click Properties.

4 In the Sheet Set Properties dialog box, click the Edit Custom Properties button.

5 In the Custom Properties dialog box, do *one* of the following:

- Click Add to create a new custom property for the sheet set. Click the name and the default value to enter the information for these fields. To assign this property to the sheet set, make sure that Sheet Set is selected under Owner.
- Click the name and the default value to change the information for these fields.
- Click a custom property from the list. Click Delete to remove the property.

6 Click OK to close each dialog box.

Toolbar: Standard
Command entry: SHEETSET

To create a custom property for a sheet

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager.

2 In the Sheet Set Manager, open a sheet set.

3 On the Sheet List tab, right-click the sheet set node.

4 In the Sheet Set Properties dialog box, click the Edit Custom Properties button.

5 In the Custom Properties dialog box, do *one* of the following:

- Click Add to create a new custom property for the sheet. Click the name and the default value to enter the information for these fields. To assign this property to the sheet, make sure that Sheet is selected under Owner.

- Click the name and the default value to change the information for these fields. With this method, you can change the value for every sheet in the sheet set.
- Click a custom property from the list. Click Delete to remove the property.

6 Click OK to close each dialog box.

Toolbar: Standard 
Command entry: SHEETSET

Quick Reference

Commands

NEWSHEETSET

Creates a new sheet set data file that manages drawing layouts, file paths, and project data.

OPENSHEETSET

Opens a selected sheet set.

SHEETSET

Opens the Sheet Set Manager.

SHEETSETHIDE

Closes the Sheet Set Manager.

System Variables

SSFOUND

Displays the sheet set path and file name if a search for a sheet set is successful.

SSLOCATE

Controls whether the sheet set associated with a drawing is located and opened when the drawing is opened.

SSMAUTOOPEN

Controls the display behavior of the Sheet Set Manager when a drawing associated with a sheet is opened.

SSMPOLLTIME

Controls whether plotting a sheet set, multi-sheet plot file, or plot spool file can be interrupted by other plot jobs.

SSMSHEETSTATUS

Controls how the status data in a sheet set is refreshed.

SSMSTATE

Indicates whether the Sheet Set Manager window is open or closed.

Utilities

No entries

Command Modifiers

No entries

Publish, Transmit, and Archive Sheet Sets

After you have organized drawings into a sheet set, you can publish, transmit, and archive the sheet set as a package.

- **Publish a sheet set:** Use the Publish feature to output the sheet set to a plotter in either normal or reverse order. For more information, see Publish a Sheet Set on page 1885. You can create single and multiple-sheet DWF or DWFx files from a sheet set or portion of a sheet set.
  For information on DWF files, see Plot DWF Files on page 1851.

  For information on DWFx files, see Plot DWFx Files on page 1852.

- **Set options for properties to include in a published DWF or DWFx file:** You can decide what types of information to reveal in your published DWF or DWFx files. The types of metadata that you can include are sheet and sheet set properties, block properties and attributes, dynamic block properties and attributes, and properties contained in custom objects. Metadata is only included when you publish to DWF or DWFx; it is not available when plotting to DWF or DWFx.

- **Transmit a sheet set:** Package and send a sheet set or a portion of a sheet set over the Internet. For more information, see Package a Set of Files for Internet Transmission.
- **Archive a sheet set:** Package a sheet set or portion of a sheet set for storage. This is very similar to packaging a transmittal set, except that you specify a folder for the archive and you do not transmit the package. For more information, see the ARCHIVE command.

Use Page Setups

Page setups provide the settings that are used for publishing and plotting. When you create a sheet set, you specify a drawing template (DWT) file that contains one or more page setups for all new sheets. This DWT file is called the *sheet creation template*.

Another DWT file, called the *page setup overrides* file, contains page setups that can be specified to override the page setups in each sheet. You specify the page setup overrides file in the Sheet Set Properties dialog box.

When you publish a sheet set, you can use the page setups defined in each drawing file, you can use the page setup overrides for all drawing files, or you can publish to a DWF or DWFx file. For more information about page setups, see Use Named Page Setups with Sheet Sets on page 1780.

NOTE With page setup overrides, you can use the PUBLISHCOLLATE system variable to control whether plotting a sheet set can be interrupted by other plot jobs or not.

Save a Selection of Sheets

You can select part of a sheet set for publishing and transmitting. On the Sheet List tab, you can select individual sheets using standard Microsoft® Windows® selection methods—press CTRL or SHIFT when you click items. You can specify all the sheets in a sheet subset by clicking the subset node.

You will likely need to perform operations on the same group of sheets in a sheet set repeatedly. To speed up selecting the sheets and to ensure that the same sheets are selected each time, you can use the Sheet Set Manager to reselect groups of sheets by name. These named groups of sheets are called *sheet selections*. You can create and manage sheet selections by using the Sheet Selections button at the top of the Sheet List tab.

To save a sheet selection

1. Click View tab ➤ Palettes panel ➤ Sheet Set Manager.
2. In the Sheet Set Manager, open a sheet set.
3. On the Sheet List tab, click the sheets and subsets to include in the sheet selection.

   You can use CTRL or SHIFT to specify several items from the list.
4. In the Sheet List Manager, near the upper-right corner, click the Sheet Selections button. Click Create.
5. In the New Sheet Selection dialog box, enter the name of the sheet selection. Click OK.

Toolbar: Standard
Command entry: SHEETSET

To restore a sheet selection

1. Click View tab ➤ Palettes panel ➤ Sheet Set Manager.
2. In the Sheet Set Manager, open a sheet set.
3. On the Sheet List tab, near the upper-right corner, click the Sheet Selections button. Select the name of the sheet selection that you want to restore.

   The sheet selection is activated and available for publishing and transmitting operations.

Toolbar: Standard
Command entry: SHEETSET

To rename or delete a sheet selection

1. Click View tab ➤ Palettes panel ➤ Sheet Set Manager.

2 In the Sheet Set Manager, open a sheet set.

3 On the Sheet List tab, near the upper-right corner, click the Sheet Selections button. Click Manage.

4 In the Sheet Selections dialog box, select the name of a sheet selection and do *one* of the following:

   - Click Rename to rename the sheet selection. Enter the new name for the sheet selection.
   - Click Delete to remove the sheet selection name from the list. Click Yes to confirm that you want to remove this sheet selection name.

5 Click OK.

Toolbar: Standard
Command entry: SHEETSET

To modify an existing sheet selection

NOTE You cannot modify a sheet selection directly. Instead, you activate a sheet selection and modify it, delete the name of the sheet selection, and then create a new sheet selection in its place.

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1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager.

2 In the Sheet Set Manager, open a sheet set.

3 On the Sheet List tab, near the upper-right corner, click the Sheet Selections button. Select the name of the sheet selection to modify.

4 Use the CTRL key to add and remove items from the sheet selection.

5 On the Sheet List Manager, near the upper-right corner, click the Sheet Selections button. Click Manage.

6 Click Delete to remove the sheet selection name from the list. Click Yes to confirm that you want to remove this sheet selection name.

7 Click OK to close the Sheet Selections dialog box.

8 On the Sheet List Manager, near the upper-right corner, click the Sheet Selections button. Click Create.

9 In the New Sheet Selection dialog box, enter the name of the sheet selection. Click OK.

Toolbar: Standard
Command entry: SHEETSET

To publish a DWF or DWFx file including layer, block, sheet set, and sheet information

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager.

2 In the Sheet Set Manager, Sheet List tab, select the sheet set to publish in a DWF or DWFx file.

3 Click the Publish button. Click Sheet Set Publish Options.

4 In the Sheet Set Publish Options dialog box, under DWF Data Options, depending on what you want to include in the published DWF or DWFx file, click any of the following to change the option to "Include."

- Layer information
- Sheet set information (attributes you can choose to include are description and custom properties)
- Sheet information (attributes you can choose to include are sheet title, sheet number, description, sheet set, subset, and sheet custom properties)
- Block information

NOTE Some sheet properties are always published in the DWF or DWFx file (no matter what setting you make in the Sheet Set Publish Options dialog box). These are: sheet name, sheet size, author, creator, creation time, and modification time.

5 Click OK.

Toolbar: Standard
Command entry: SHEETSET

To publish a DWF or DWFx file using a block template file to include block properties and attributes

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager. 

2 In the Sheet Set Manager, Sheet List tab, select the sheet set to publish in a DWF or DWFx file.

3 Click the Publish button. Click Sheet Set Publish Options.

4 In the Sheet Set Publish Options dialog box, under DWF Data Options, Block Information, click to display the drop-down list, and select “Include.”

5 Under DWF Data Options, Block Template File, select the block template file you want to use or create a new one. Click OK.

   The block template file defining the blocks and their properties and attributes will be included in the published DWF or DWFx file.

Toolbar: Standard 
Command entry: SHEETSET

Quick Reference

Commands

ARCHIVE

Packages the current sheet set files for archive.

NEWSHEETSET

Creates a new sheet set data file that manages drawing layouts, file paths, and project data.

OPENSHEETSET

Opens a selected sheet set.

SHEETSET

Opens the Sheet Set Manager.

SHEETSETHIDE

Closes the Sheet Set Manager.

System Variables

PUBLISHCOLLATE

Controls whether plotting a sheet set, multi-sheet plot file, or plot spool file can be interrupted by other plot jobs.

SSFOUND

Displays the sheet set path and file name if a search for a sheet set is successful.

SSLOCATE

Controls whether the sheet set associated with a drawing is located and opened when the drawing is opened.

SSMAUTOOPEN

Controls the display behavior of the Sheet Set Manager when a drawing associated with a sheet is opened.

SSMSHEETSTATUS

Controls how the status data in a sheet set is refreshed.

SSMSTATE

Indicates whether the Sheet Set Manager window is open or closed.

Utilities

No entries

Command Modifiers

No entries

Use Sheet Sets in a Team

You can use sheet sets in a team that can involve network access, Internet collaboration, and email transmittal. The team can also include people who use software that does not include Sheet Set Manager.

Work in a Team That Uses Sheet Set Manager

When you use sheet sets in a team, each member should have network access to the sheet set data (DST) file and the drawing template (DWT) files associated with the sheet set. Each team member can open the sheet set to load the sheet set information from the DST file into the Sheet Set Manager.

Any changes that any team member makes opens the DST file briefly and updates the information stored in that file. When the DST file is opened, a lock icon is displayed next to the sheet set name in the top left corner of the Sheet Set Manager.

A green dot in the lock icon indicates that the Sheet Set Manager session on your computer has temporarily locked the DST file.

A red dot indicates that the Sheet Set Manager session on a team member's computer has temporarily locked the DST file.

A yellow dot in the lock icon means that the sheet is in a special state; for example, its file properties may be set to Read-Only.

Other members of the team can automatically see changes to the sheet set in the Sheet Set Manager tree view.

If each member of the team has access to the sheet set DWT files, new drawing files and their sheets are created using the same drawing template file; page setups for these drawings are also standardized.

NOTE If two or more users access the same sheet files through different logical drives on a network, each will in turn be prompted to resave the sheet set using their own logical drive. To avoid unnecessary saving, users should map the same logical drives, if possible.

Status data for sheets in the current sheet set is also available to other team members. This status data is displayed in the tree view and indicates one of the following conditions:

 The sheet is available for editing.

 The sheet is locked.

 The sheet is missing or found in an unexpected folder location.

The active sheets of other team members are automatically polled for status changes; the tree view is updated in your session of the Sheet Set Manager.

The polling cycle skips the poll interval in your session when a command is active. To force a sheet status update, click Refresh Sheet Status on the Sheet List tab.

You can click any sheet to display more information in the Details area of the Sheet Set Manager.

NOTE A false lock icon may be displayed if there is a network problem or if the program terminates unexpectedly. If you suspect a problem, click the sheet to display more information.

Work in a Team That Does Not Use Sheet Set Manager

With some limitations, you can use sheet sets in a team with members who do not have network access, or do not have access to the Sheet Set Manager. These team members may be using AutoCAD LT or an older version of AutoCAD. In those circumstances, not all members of the team will have access to the DST file. However, relevant information from the DST file is stored (cached) in each drawing file, and sheet set information, such as custom properties, is preserved when the drawing file is shared by other team members.

After a member of the team changes information in the DST file, the information in several drawing files might need to be updated. With the sheet set open, update a sheet by opening and saving the sheet.

You can update all sheets in a sheet set automatically with the Resave All Sheets option in the sheet set shortcut menu. Drawing files saved in a previous DWG file format are saved without changing the format.

NOTE In a network environment, make sure that all drawing files used in the current sheet set that are opened by other users are closed before performing the Resave All Sheets operation.

To resave all sheets in a sheet set

1. Click View tab ➤ Palettes panel ➤ Sheet Set Manager. 
2. In the Sheet Set Manager, open a sheet set.
3. On the Sheet List tab, right-click the sheet set node. Click Resave All Sheets.

   All sheets in the sheet set are resaved. This updates the sheet set information that is stored in each drawing file.

Toolbar: Standard 
Command entry: SHEETSET

Quick Reference

Commands

NEWSHEETSET

Creates a new sheet set data file that manages drawing layouts, file paths, and project data.

OPENSHEETSET

Opens a selected sheet set.

SHEETSET

Opens the Sheet Set Manager.

SHEETSETHIDE

Closes the Sheet Set Manager.

System Variables

SSFOUND

Displays the sheet set path and file name if a search for a sheet set is successful.

SSLOCATE

Controls whether the sheet set associated with a drawing is located and opened when the drawing is opened.

SSMAUTOOPEN

Controls the display behavior of the Sheet Set Manager when a drawing associated with a sheet is opened.

SSMPOLLTIME

Controls whether plotting a sheet set, multi-sheet plot file, or plot spool file can be interrupted by other plot jobs.

SSMSHEETSTATUS

Controls how the status data in a sheet set is refreshed.

SSMSTATE

Indicates whether the Sheet Set Manager window is open or closed.

Utilities

No entries

Command Modifiers

No entries

Create and Modify Objects

Control the Properties of Objects

15

You can organize objects in your drawing and control how they are displayed and plotted by changing their properties, which include layer, linetype, color, lineweight, and plot style.

Layers are like transparent overlays on which you organize and group different kinds of drawing information. The objects you create have common properties including colors, linetypes, and lineweights. An object can assume these properties from the layer it is drawn on, or properties can be specifically assigned to individual objects. Color helps you distinguish similar elements in your drawings, while linetypes help you differentiate easily between different drafting elements. Lineweights represent the size or type of an object through width, enhancing your drawing and increasing legibility. Organizing layers and the objects on layers makes it easier to manage the information in your drawings.

Work with Object Properties

You can change the object properties in your drawing by using the Properties palette or the Quick Properties palette.

Overview of Object Properties

Every object you draw has properties. Some properties are general and apply to most objects; for example, layer, color, linetype, and plot style. Other properties are object-specific; for example, the properties of a circle include radius and area, and the properties of a line include length and angle.

Most general properties can be assigned to an object by layer or can be assigned to an object directly.

- When a property is set to the value BYLAYER, the object is assigned the same value as the layer on which it is drawn.

For example, if a line drawn on layer 0 is assigned the color BYLAYER, and layer 0 is assigned the color Red, the line is red.

- When a property is set to a specific value, that value overrides the value set for the layer.
  For example, if a line drawn on layer 0 is assigned the color Blue, and layer 0 is assigned the color Red, the line is blue.

See also:

- Control the Color and Linetype Properties in Blocks on page 860

Quick Reference

Commands

PROPERTIES

Controls properties of existing objects.

PROPERTIESCLOSE

Closes the Properties palette.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Display and Change the Properties of Objects

You can display and change the current properties for any object in your drawing.

You can display and change the current properties for any object in your drawing in the following ways:

- Open the Quick Properties palette to view and change the settings for selected properties of the object.
- Open the Properties palette and view and change the settings for all properties of the object.
- View and change the settings in the Layer control on the Layers toolbar and the Color, Linetype, Lineweight, and Plot Style controls on the Properties toolbar.
- Use the LIST command to view information in the text window.
- Use the ID command to display a coordinate location.

Use the Quick Properties Palette

The Quick Properties palette lists the most commonly used properties for each object type or a set of objects. You can easily customize the quick properties for any object in the Customize User Interface (CUI) editor. See Quick Properties in the *Customization Guide*.

- When one or more objects of the same type are selected, the Quick Properties palette displays the selected properties of that object type.
- When two or more objects of different types are selected, the Quick Properties palette displays the common properties, if any for all objects in the selection set.

You can select any object to display the Quick Properties palette when the QPMODE system variable is set to 1. When the QPMODE system variable is set to 2, the Quick Properties palette displays only if the selected object is defined in the Customize User Interface (CUI) editor to display properties. You can use the QPLOCATION system variable to display the Quick Properties palette by using cursor or float mode. You can also control the display settings of the Quick Properties palette using the Drafting Settings Dialog Box.

Use the Properties Palette

The Properties palette lists the current settings for properties of the selected object or set of objects. You can modify any property that can be changed by specifying a new value.

- 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, the name of the plot style table attached to the layer, the view properties, and information about the UCS.

You can double-click most objects to open the Properties palette when the DBLCLKEDIT system variable is set to On (the default). The exceptions include blocks and attributes, hatches, gradient fills, text, multilines, and xrefs. If you double-click any of these objects, an object-specific dialog box is displayed instead of the Properties palette.

NOTE The DBLCLKEDIT system variable must be set to On and the PICKFIRST system variable must be set to 1(the default) for double-clicking to work.

Change Object Property or ByBlock Settings to ByLayer

Using the SETBYLAYER command, you can change specified properties to ByLayer for selected objects. Objects that have a ByBlock setting can also be changed to ByLayer. When an object's properties are not set to ByLayer, those objects do not display the layer property overrides that were set by viewport.

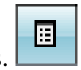
In the SetByLayer Settings dialog box, you can specify which object property settings are changed to ByLayer.

The SETBYLAYERMODE system variable stores the property settings that are to be changed when the SETBYLAYER command is used.

See also:

- Control the Color and Linetype Properties in Blocks on page 860
- Filter Selection Sets on page 1066
- Set Interface Options on page 111
- Quick Properties in the *Customization Guide*

To turn on and turn off the Quick Properties palette

- On the status bar, click Quick Properties. 
  If you want to dismiss the Quick Properties palette temporarily, press ESC.

To disable the Quick Properties palette

1. In the Quick Properties palette, click the Options button.
2. Click Disable.

NOTE When you select the Disable option, the value of QPMODE system variable is changed to a negative number and the feature is turned off.

To change the settings of a Quick Properties palette

1. Click Tools menu ➤ Drafting Settings.
2. In the Drafting Settings dialog box, Quick Properties tab, select Quick Properties On.
3. Select the location mode for the Quick Properties palette.
4. Under Size Settings, select or clear the Auto-Collapse option to expand or collapse the Quick Properties palette. If this option is selected, enter the height value in the text box.
5. Click OK.

Command entry: DSETTINGS

To set the Quick Properties palette to always display at its full height

1. In the Quick Properties palette, click the Options button.
2. Clear the Auto-Collapse option.

To display the properties of a single object

1. Select the object.
2. Click View tab ➤ Palettes panel ➤ Properties.
   The Properties palette displays the properties of the selected object.
3. Alternatively, you can right-click in the drawing and then, click Properties.

NOTE You can also double-click almost any object to display the Properties palette.

Toolbar: Standard 
Command entry: PROPERTIES

To list database information for several objects

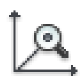
1 Click Home tab ➤ Properties panel ➤ List. 

2 Select one or more objects and press ENTER.

The text window displays a report.

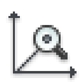
Toolbar: Inquiry 
Command entry: LIST

To display the coordinate values of a point

1 Click Tools tab ➤ Inquiry panel ➤ ID Point. 

2 Specify the point whose coordinate values you want to identify.

The *X*, *Y*, and *Z* values are displayed at the Command prompt.

Toolbar: Inquiry 
Command entry: ID

To change properties of objects in the Properties palette

1 Select one or more objects.

2 Click View tab ➤ Palettes panel ➤ Properties.

The Properties palette displays the properties of the selected object.

3 In the Properties palette, use the scroll bar next to the title bar to scroll through the list of properties. You can click the arrow at the right of each category to expand or collapse the list.

4 Select the value you want to change; use one of the following methods to change the value:

- 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.

Changes are applied immediately.

5 To undo a change, right-click an empty area in the Properties palette. Click Undo.

6 Press ESC to remove the selection.

Toolbar: Standard 
Command entry: PROPERTIES

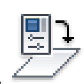
To change an object's property setting to ByLayer

1 Click Home tab ➤ Modify panel ➤ Set to ByLayer. 

2 Enter **S** to specify which properties to change to ByLayer or select objects using a standard selection method and press ENTER.

Command entry: SETBYLAYER

To change an object's ByBlock setting to ByLayer

1 Click Home tab ➤ Modify panel ➤ Set to ByLayer . 

2 Select objects using a standard selection method and press ENTER.

3 When prompted, enter **Y** to change ByBlock to Bylayer.

4 When prompted, specify whether blocks are included.

Command entry: SETBYLAYER

Quick Reference

Commands

CUI

Manages the customized user interface elements in the product.

DSETTINGS

Sets grid and snap, polar and object snap tracking, object snap modes, Dynamic Input, and Quick Properties.

ID

Displays the UCS coordinate values of a specified location.

LIST

Displays property data for selected objects.

PROPERTIES

Controls properties of existing objects.

PROPERTIESCLOSE

Closes the Properties palette.

SETBYLAYER

Changes the property overrides of selected objects to ByLayer.

System Variables

DBLCLKEDIT

Controls the double click editing behavior in the drawing area.

LUPREC

Sets the display precision for linear units and coordinates.

OPMSTATE

Indicates whether the Properties palette is open, closed, or hidden.

PALETTEOPAQUE

Controls whether palettes can be made transparent.

PICKFIRST

Controls whether you select objects before (noun-verb selection) or after you issue a command.

QPLOCATION

Sets the location mode of Quick Properties palette.

QPMODE

Sets the on or off state of Quick Properties palette.

SETBYLAYERMODE

Controls which properties are selected for the SETBYLAYER command.

Utilities

No entries

Command Modifiers

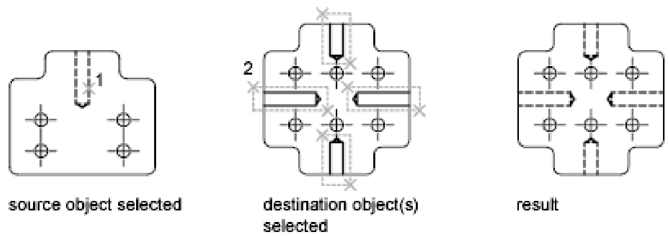
No entries

Copy Properties Between Objects

You can copy some or all properties of one object to other objects using Match Properties.

The types of properties that can be copied include, but are not limited to, color, layer, linetype, linetype scale, lineweight, plot style, viewport property overrides, and 3D thickness.

By default, all applicable properties are automatically copied from the first object you selected to the other objects. If you don't want a specific property or properties to be copied, use the Settings option to suppress the copying of that property. You can choose the Settings option at any time during the command.



To copy properties from one object to other objects

1. Click Home tab ➤ Properties panel ➤ Match Properties. 
2. Select the object whose properties you want to copy.
3. If you want to control which properties are transferred, enter **s** (Settings). In the Property Settings dialog box, clear the items that you do not want copied (all are on by default). Click OK.
4. Select the objects to which you want to apply the selected properties and press ENTER.

Toolbar: Standard 
Command entry: MATCHPROP

Quick Reference

Commands

MATCHPROP

Applies the properties of a selected object to other objects.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

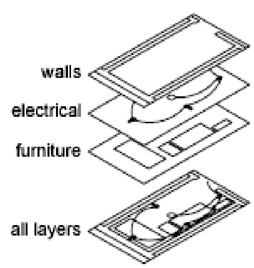
Work with Layers

Layers are like transparent overlays on which you organize and group objects in a drawing.

Overview of Layers

Layers are used to group information in a drawing by function and to enforce linetype, color, and other standards.

Layers are the equivalent of the overlays used in paper-based drafting. Layers are the primary organizational tool used in drawing. You use layers to group information by function and to enforce linetype, color, and other standards.



By creating layers, you can associate similar types of objects by assigning them to the same layer. For example, you can put construction lines, text, dimensions, and title blocks on separate layers. You can then control the following:

- Whether objects on a layer are visible or dimmed in any viewports
- Whether and how objects are plotted
- What color is assigned to all objects on a layer

- What default linetype and lineweight are assigned to all objects on a layer
- Whether objects on a layer can be modified
- Whether objects display with different layer properties in individual layout viewports

Every drawing includes a layer named 0. Layer 0 cannot be deleted or renamed. It has two purposes:

- Ensure that every drawing includes at least one layer
- Provide a special layer that relates to controlling colors in blocks

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NOTE It is recommended that you create several new layers with which to organize your drawing rather than create your entire drawing on layer 0.

Quick Reference

Commands

CLASSICLAYER

Opens the modal Layer Properties Manager.

LAYER

Manages layers and layer properties.

LAYERPALETTE

Opens the modeless Layer Properties Manager.

System Variables

LAYERDLGMODE

Sets the flavor of the Layer Properties Manager that is defined for the use of the LAYER command.

SHOWLAYERUSAGE

Displays icons in the Layer Properties Manager to indicate whether layers are in use.

Utilities

No entries

Command Modifiers

No entries

Use Layers to Manage Complexity

You can use layers to control the visibility of objects and to assign properties to objects. Layers can be locked to prevent objects from being modified.

You can reduce the visual complexity of a drawing and improve display performance by controlling how objects are displayed or plotted. For example, you can use layers to control the properties and visibility of similar objects, such as electrical parts or dimensions. Also, you can lock a layer to prevent objects on that layer from being accidentally selected and modified.

Control the Visibility of Objects on a Layer

You can make drawing layers invisible either by turning them off or by freezing them. Turning off or freezing layers is useful if you need an unobstructed view when working in detail on a particular layer or set of layers or if you don't want to plot details such as reference lines. Whether you choose to freeze layers or turn them off depends on how you work and on the size of your drawing.

- **On/Off**. Objects on turned-off layers are invisible, but they still hide objects when you use HIDE. When you turn layers on and off, the drawing is not regenerated.
- **Freeze/Thaw.** Objects on frozen layers are invisible and do not hide other objects. In large drawings, freezing unneeded layers speeds up operations involving display and regeneration. Thawing one or more layers may cause the drawing to be regenerated. Freezing and thawing layers takes more time than turning layers on and off.

In a layout, you can freeze layers in individual layout viewports.

NOTE Instead of turning off or freezing a layer, you can fade the layer by locking it. See “Lock the Objects on a Layer” below.

Assign a Default Color and Linetype to a Layer

Each layer has associated properties such as color and linetype that are assumed by all objects on that layer when the setting is ByLayer. For example, if the Color control on the Properties toolbar is set to BYLAYER, the color of new objects is determined by the color setting for the layer in the Layer Properties Manager.

If you set a specific color in the Color control, that color is used for all new objects, overriding the default color for the current layer. The same is true for the Linetype, Lineweight, and Plot Style controls on the Properties toolbar.

The BYBLOCK setting should be used only for creating blocks. See Control the Color and Linetype Properties in Blocks on page 860.

Override Layer Properties in a Layout Viewport

Some layer properties can be changed using overrides on a viewport basis in layouts. Using layer property overrides is an efficient way to display objects with different property settings for color, linetype, lineweight, and plot style. Layer property overrides are applied to the current layout viewport.

For example, if you want objects on the Electrical layer to display prominently in one of two layout viewports, you set a Color override on the Electrical layer for each of the two viewports. By setting the color red for one viewport and gray for the other, you easily accomplish this objective without changing the global color property assigned to the layer. See Override Layer Properties in Viewports on page 559 for more information.

Lock the Objects on a Layer

When a layer is locked, none of the objects on that layer can be modified until you unlock the layer. Locking layers reduces the possibility of modifying objects accidentally. You can still apply object snaps to objects on a locked layer and perform other operations that do not modify those objects.

You can fade the objects on locked layers to make them appear more faint than other objects. This serves two purposes:

- You can easily see what objects are on locked layers.
- You can reduce the visual complexity of a drawing but still maintain visual reference and object snapping capabilities to those objects.

The LAYLOCKFADECTL system variable controls the fading applied to locked layers. Locked layers that are faded are plotted normally.

NOTE Grips are not displayed on objects that are on locked layers.

To make a selected layer the current layer

1 Click Home tab ➤ Layers panel ➤ Make Object's Layer Current.

2 Select an object on the layer you want to make current.

Toolbar: Layers
Command entry: LAYMCUR

To copy an object to another layer

1 Click Home tab ➤ Layers panel ➤ Copy Objects to New Layer.

2 Select the objects you want to copy.

3 Press ENTER.

4 Select an object on the layer where you want the copied object to be placed.

5 Do one of the following:
   - Specify a base point, and then specify a second point for the location of the object on the destination layer.
   - Enter the displacement in the form of a Cartesian, polar, cylindrical, or spherical coordinate value. At the prompt for the second point of displacement, press ENTER.

Toolbar: Layers II
Command entry: COPYTOLAYER

To dynamically display layers

1 Click Home tab ➤ Layers panel ➤ Layer Walk.

2 In the LayerWalk dialog box, click a layer to view the objects on that layer. To select more than one layer, press CTRL and click the layers you want to view.

   The drawing displays the objects on the layers you selected.

3 Click Close.

Toolbar: Layers II 
Command entry: LAYWALK

To filter a layer list

1 Click Home tab ➤ Layers panel ➤ Layer Walk. 

2 In the LayerWalk dialog box, Filter list, do one of the following:
   - Enter an existing filter name or select a filter from the Filter drop-down list.
   - Enter filter criteria (using wild-card characters) to create a list of the layers you want filtered. For example, if want to filter layers named “0” through “3”, you could enter a filter criteria of [0-3]*.

   NOTE For more information about wild-card characters, see the “Wild-Card Characters” section of Filter and Sort the List of Layers on page 567.

   The layer list displays layers defined in the filter.

3 Click Close.

Toolbar: Layers II 
Command entry: LAYWALK

To create and save a layer filter in the LayerWalk dialog box

1 Click Home tab ➤ Layers panel ➤ Layer Walk. 

2 In the LayerWalk dialog box, Filter list, enter filter criteria (using wild-card characters) to create a list of the layers you want filtered. For example, if want to filter layers “0” through “3”, enter a filter criteria of [0-3]*.

NOTE For more information about wild-card characters, see the “Wild-Card Characters” section of Filter and Sort the List of Layers on page 567.

3 Right-click, and then click Save Current Filter.

4 Click Close.

Toolbar: Layers II
Command entry: LAYWALK

To turn on or off an active layer filter

1 Click Home tab ➤ Layers panel ➤ Layer Walk.

2 In the LayerWalk dialog box, Filter list, enter an existing filter name or select a filter from the Filter drop-down list.

3 Click the Filter check box to display or hide the layers in the layer list that are defined in the active filter. The display of the layers in the drawing itself does not change.

4 Click Close.

Toolbar: Layers II
Command entry: LAYWALK

To turn the Always Show option on or off in the LayerWalk dialog box

1 Click Home tab ➤ Layers panel ➤ Layer Walk.

2 In the LayerWalk dialog box, Layer list, double-click the layer you want to set to Always Show. To select more than one layer, press SHIFT and double-click the additional layers.

An asterisk (*) is displayed to the left of each layer that you set to Always Show. These layers are displayed in the drawing regardless of whether or not they are in an active filter.

3 Click Close.

Toolbar: Layers II
Command entry: LAYWALK

To purge unreferenced layers using the LayerWalk dialog box

1 Open a drawing that contains one or more unreferenced layers that you want to purge.

2 Click Home tab ➤ Layers panel ➤ Layer Walk.

3 In the LayerWalk dialog box, right-click. Click Select Unreferenced.
   All unreferenced layers are selected.

   NOTE The Purge button is active only when the layers you select are unreferenced.

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4 Click Purge.

5 Click Close.

NOTE You can also purge layers in the Layer Properties Manager. See To Remove an Unused Layer. on page 552

Command entry: LAYWALK

Toolbar: Layers II

To move an object to the current layer

1 Click Home tab ➤ Layers panel ➤ Change to Current Layer.

2 Select the objects you want to move to the current layer.

3 Press ENTER.


Toolbar: Layers II
Command entry: LAYCUR

To delete a layer by selecting an object on that layer

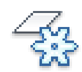
1 Click Home tab ➤ Layers panel ➤ Delete. 

2 Select an object on the layer you want to delete.

3 Press ENTER.
All objects on the selected layer and the layer itself are deleted.

Command entry: LAYDEL

To freeze a layer by selecting an object on that layer

1 Click Home tab ➤ Layers panel ➤ Freeze. 

2 Select an object on the layer you want to freeze,.

3 Press ENTER.
The selected layer is frozen.


Toolbar: Layers II
Command entry: LAYFRZ

To thaw all layers

- Click Home tab ➤ Layers panel ➤ Thaw All Layers. 
All layers (except layers that are frozen in a viewport) are thawed.

Command entry: LAYTHW

To display selected layers while turning off all other layers

1. Click Home tab ➤ Layers panel ➤ Isolate. 
2. Select an object on the layer you want to isolate.
3. Press ENTER.

   The selected layer is isolated.

   NOTE To restore layers to the layer state before you isolated them, use the LAYUNISO command. Any layer settings you changed are preserved.

Toolbar: Layers II 
Command entry: LAYISO

To restore layer settings when exiting the LayerWalk dialog box

1. Click Home tab ➤ Layers panel ➤ Walk. 
2. In the LayerWalk dialog box, click a layer to view the objects on that layer. To select more than one layer, click a layer in the list and drag the cursor over the layers you want to view.
3. Do one of the following:
   - To return to the original view when you close the dialog box, make sure the Restore on Exit check box is selected.
   - To return to the drawing with only the selected layers, make sure the Restore on Exit check box is blank.
4. Click Close.

Toolbar: Layers II 
Command entry: LAYWALK

To restore a drawing's layer states after isolating a layer

- After isolating a layer, and (optionally) making other layer changes, click Home tab ➤ Layers panel ➤ Unisolate. 
  The drawing is restored to the layer state before the layers were isolated.

Toolbar: Layers II 
Command entry: LAYUNISO

To isolate an object's layer to the current viewport

1 Click a layout tab.

2 Click Home tab ➤ Layers panel ➤ Isolate to Current Viewport. 

3 Select the object whose layer you want to isolate. Press ENTER.
The selected layer is frozen in all viewports except the current viewport.

Command entry: LAYVPI

To turn a layer on and off by using the Layers toolbar

1 Click Home tab ➤ Layers panel ➤ Layer Properties 

2 In the Layer Properties Manager, click the light bulb for the layer name that you want to turn on or off.
If the light bulb is yellow, the layer is turned on.

Toolbar: Layers 
Command entry: LAYER

To turn all layers on

- Click Home tab ➤ Layers panel ➤ Turn All Layers On. 

All layers in the drawing are turned on.

Command entry: LAYON

To turn a layer off by selecting an object on the layer

1. Click Home tab ➤ Layers panel ➤ Off.
2. Select the object whose layer you want to turn off. Press ENTER.

Toolbar: Layers II
Command entry: LAYOFF

To copy properties from one layer to other layers

1. Click Home tab ➤ Layers panel ➤ Match.
2. Select the object whose layer you want to change. Press ENTER.
3. Select an object on the layer where you want the object to move.

   The object is moved to the selected layer.

Toolbar: Layers II
Command entry: LAYMCH

To merge selected layers onto a destination layer

1. Click Home tab ➤ Layers panel ➤ Merge.
2. Select any object on the layer you want to delete. Press ENTER.
3. Select any object on the layer where you want all the objects from the first layer to be merged.

   The objects from the first layer are moved to the selected layer, and the first layer is deleted.

Command entry: LAYMRG

To assign a color to a layer

1 Click Home tab ➤ Layers panel ➤ Layer Properties
2 In the Layer Properties Manager, select a layer. Click the color icon.
3 In the Select Color dialog box, select a color.
4 Click Apply to save your changes, or click OK to save and close.

Toolbar: Layers
Command entry: LAYER

To change the properties of more than one layer

1 Click Home tab ➤ Layers panel ➤ Layer Properties
2 In the Layer Properties Manager list view, use one of the following methods to select the layers:
   - Hold down CTRL and select layer names.
   - Right-click. Click Show Filters in Layer List to display a check mark, and then select a layer filter.
3 Click the icons for the properties you want to change.
4 Click Apply to save your changes, or click OK to save and close.

Toolbar: Layers
Command entry: LAYER

To hide the tree view in the Layer Properties Manager

- Click Home tab ➤ Layers panel ➤ Layer Properties

- In the Layers Properties Manager, right-click in the list view. Clear the Show Filter Tree option.

To move a column in the Layer Properties Manager

- Click Home tab ➤ Layers panel ➤ Layer Properties 
- In the Layer Properties Manager, click on any column name and drag it to a new location.
- Right-click on any column name and click Customize. In the Customize Layer Columns dialog, highlight the columns to move and click Move Up to move the columns to the left in the Layer Properties Manager. Click Move Down to move the columns to the right in the Layer Properties Manager.

NOTE Column settings are specific to the drawing space in which they were set. For example, column settings applied in model space are not applied to the Layer Properties Manager when opened from paper space.

Toolbar: Layers 
Command entry: LAYER

To hide a column in the Layer Properties Manager

- Click Home tab ➤ Layers panel ➤ Layers Properties 
- In the Layer Properties Manager, right-click on any column name and select the column that is checked from the shortcut menu.
- Right-click on any column name and click Customize. In the Customize Layer Columns dialog, uncheck the columns to hide.

Columns names that are unchecked in the shortcut menu are hidden. To show the column, right-click any column name and select the column name from the shortcut menu. The Name column cannot be hidden.

NOTE Column settings are specific to the drawing space in which they were set. For example, column settings applied in model space are not applied to the Layer Properties Manager when opened from paper space.

Toolbar: Layers

Command entry: LAYER

To reorder columns in the Layer Properties Manager

- Click Home tab ➤ Layers panel ➤ Layer Properties
- In the Layer Properties Manager, click on the column name and drag and release to the desired location.
- Right-click on any column name and click Customize. In the Customize Layer Columns dialog, check the column or columns to move and click Move Up or Move Down.

Toolbar: Layers

Command entry: LAYER

To reset all columns to their default settings in the Layer Properties Manager

- Click Home tab ➤ Layers panel ➤ Layer Properties
- In the Layer Properties Manager, right-click on any column name and select Restore all Columns to Defaults from the shortcut menu.

All columns are reset to their original position, column width, and display status.

NOTE Column settings are specific to the drawing space in which they were set. For example, column settings applied in model space are not applied to the Layer Properties Manager when opened from paper space.

Toolbar: Layers

Command entry: LAYER

To lock or unlock a layer by using the Layers toolbar

1 Click Home tab ➤ Layers panel ➤ Layers Properties

2 In the Layer Properties Manager, click the padlock icon for the layer name that you want to lock or unlock.

   If the padlock is open, the layer is unlocked.

3 (Optional) Specify the degree of fading for all locked layers with the LAYLOCKFADECTL system variable.

Toolbar: Layers
Command entry: LAYER, LAYLOCKFADECTL

Quick Reference

Commands

CLASSICLAYER

Opens the modal Layer Properties Manager.

COPYTOLAYER

Copies one or more objects to another layer.

LAYCUR

Changes the layer property of selected objects to the current layer.

LAYDEL

Deletes all objects on a layer and purges the layer.

LAYER

Manages layers and layer properties.

LAYERPALETTE

Opens the modeless Layer Properties Manager.

LAYFRZ

Freezes the layer of selected objects.

LAYISO

Hides or locks all layers except those of the selected objects.

LAYMCH

Changes the layer of a selected object to match the destination layer.

LAYMCUR

Sets the current layer to that of a selected object.

LAYMRG

Merges selected layers into a target layer, removing the previous layers from the drawing.

LAYOFF

Turns off the layer of a selected object.

LAYON

Turns on all layers in the drawing.

LAYTHW

Thaws all layers in the drawing.

LAYUNISO

Restores all layers that were hidden or locked with the LAYISO command.

LAYWALK

Displays objects on selected layers and hides objects on all other layers.

System Variables

LAYERDLGMODE

Sets the flavor of the Layer Properties Manager that is defined for the use of the LAYER command.

LAYLOCKFADECTL

Controls the amount of fading for objects on locked layers.

No entries

Utilities

No entries

Command Modifiers

No entries

Create and Name Layers

You can create and name a new layer for each conceptual grouping (such as walls or dimensions) and assign common properties to each layer.

By organizing objects into layers, you can control the visibility and object properties of a large number of objects separately for each layer and make changes quickly.

NOTE The number of layers that you can create in a drawing and the number of objects that you can create on each layer are practically unlimited.

Choose Layer Names Carefully

A layer name can include up to 255 characters (double-byte or alphanumeric): letters, numbers, spaces, and several special characters. Layer names cannot include the following characters:

< > / \ “ : ; ? * | = ‘

In many cases, the layer names you choose are dictated by corporate, industry, or client standards.

The Layer Properties Manager sorts layers alphabetically by name. If you organize your own layer scheme, choose layer names carefully. Use common prefixes to name layers with related drawing components, you can use wild-card characters in layer name filters when you need to find those layers quickly.

NOTE If you consistently use a specific layering scheme, you can set up a drawing template with layers, linetypes, and colors already assigned. For more information about creating templates, see Use a Template File to Start a Drawing on page 175.

Copy Layers from Another Drawing

You can use DesignCenter™ to copy layers from any drawing to another by dragging. For example, you might have a drawing that contains all the standard layers needed for a project. You can create a new drawing and use DesignCenter to drag the predefined layers to the new drawing, which saves you time and ensures consistency between drawings.

You can also drag layers or copy layers by double-clicking or by clicking Insert on the shortcut menu.

NOTE You need to resolve duplicate layer names before you drag layers from DesignCenter.

Select a Layer to Draw On

As you draw, newly created objects are placed on the current layer. The current layer may be the default layer (0) or a layer you create and name yourself. You switch from one layer to another by making a different layer current; any subsequent objects you create are associated with the new current layer and use its color, linetype, and other properties. You cannot make a layer the current layer if it is frozen or if it is an xref-dependent layer.

Remove Layers

You can remove unused layers from your drawing with PURGE or by deleting the layer from the Layer Properties Manager. 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.

WARNING 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.

To create a new layer

1 Click Home tab ➤ Layers panel ➤ Layer Properties. 

2 In the Layer Properties Manager, click the New Layer button.
A layer name, such as LAYER1, is automatically added to the layer list.

3 Enter a new layer name by typing over the highlighted layer name.

4 To change the properties, click icons.

When you click Color, Linetype, Lineweight, or Plot Style, a dialog box is displayed.

5 (Optional) Click in the Description column and enter text.

6 Click Apply to save your changes, or click OK to save and close.

Toolbar: Layer Properties 
Command entry: LAYER

To remove an unused layer

1 Click Home tab ➤ Layers panel ➤ Layer Properties. 

2 In the Layer Properties Manager, select the layer. Click the Delete Layer button.

   Layers that have objects assigned to them cannot be removed until those objects are reassigned to a different layer or are deleted. Layers 0 and DEFPOINTS and the current layer cannot be removed.

3 Click Apply to save your changes, or click OK to save and close.

   The selected layer is removed.

Toolbar: Layers 
Command entry: LAYER

To purge all unused layers

1 Click Tools tab ➤ Drawing Utilities panel ➤ Purge. 

   The Purge dialog box displays a tree view of object types with items that can be purged (removed from the drawing).

2 To purge unreferenced layers, use one of the following methods:
   - To purge all unreferenced layers, select Layers.
   - To purge specific layers, double-click Layers to expand the tree view. Select the layers to be purged.

3 If the item you want to purge is not listed, select View Items You Cannot Purge, select the layer, and read the explanation.

4 You are prompted to confirm each item in the list. If you do not want to confirm each purge, clear the Confirm Each Item to Be Purged option.

5 Click Purge.

6 To confirm the purging of each item, respond to the prompt by choosing Yes or No, or Yes to All if more than one item is selected.

7 Click Close.

Command entry: PURGE

Quick Reference

Commands

CLASSICLAYER

Opens the modal Layer Properties Manager.

LAYER

Manages layers and layer properties.

LAYERPALETTE

Opens the modeless Layer Properties Manager.

PURGE

Removes unused items, such as block definitions and layers, from the drawing.

System Variables

CLAYER

Sets the current layer.

LAYERDLGMODE

Sets the flavor of the Layer Properties Manager that is defined for the use of the LAYER command.

Utilities

No entries

Command Modifiers

No entries

Change Layer Settings and Layer Properties

You can change the name of a layer and any of its properties, including color and linetype, and you can reassign objects from one layer to another.

Because everything in your drawing is associated with a layer, it's likely that in the course of planning and creating a drawing, you'll need to change what you place on a layer or how you view the layers in combination. You can

- Reassign objects from one layer to another.
- Change the name of a layer.
- Change the default color, linetype, or other properties of the layer.

Reassigning an object to a different layer is useful if you create an object on the wrong layer or decide to change your layer organization. Unless the color, linetype, or other properties of the object have been set explicitly, an object that you reassign to a different layer will acquire the properties of that layer.

You can change layer properties in the Layer Properties Manager and in the Layer control on the Layers toolbar. Click the icons to change settings. Layer names and colors can be changed only in the Layer Properties Manager, not the Layer control.

Undo Changes to Layer Settings

You can use Layer Previous to undo changes you make to layer settings. For example, if you freeze several layers and change some of the geometry in a drawing, and then want to thaw the frozen layers, you can do this with a single command without affecting the geometry changes. In another example, if you changed the color and linetype of several layers but later decide you prefer the old properties, you can use Layer Previous to undo the changes and restore the original layer settings.

When you use Layer Previous, it undoes the most recent change or set of changes made using either the Layer control or the Layer Properties Manager. Every change you make to layer settings is tracked and can be undone with

Layer Previous. You can use LAYERPMODE to suspend layer property tracking when you don't need it, such as when you run large scripts. There is a modest performance gain in turning off Layer Previous tracking.

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 layer 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.

Changes in the Layer Properties Manager can be grouped by enabling Combine Layer Property Change option in User Preferences tab in the Options dialog box.. Layer creation and deletion will be tracked in the Undo list as unique items.

To change the layer of one or more objects

1. Select the objects whose layer you want to change.
2. Click Home tab ➤ Layers panel ➤ Layer Properties 
3. In the Layer Properties Manager, select the layer that you want to assign to the objects.
4. Press ESC to remove selection.

Toolbar: Layers 

To change the default linetype assigned to a layer

1. Click Home tab ➤ Layers Panel ➤ Layer Properties  .
2. In the Layer Properties Manager, select the linetype for the layer you want to change.

3 In the Select Linetype dialog box, if the linetype you need is not displayed, click Load and use one of the following methods:
   - In the Load or Reload Linetypes dialog box, select one or more linetypes to load. Click OK to return to the Select Linetype dialog box.
   - In the Load or Reload Linetypes dialog box, click File to open additional linetype definition (LIN) files. Select one or more linetypes to load. Click OK to return to the Select Linetype dialog box.
4 Select the linetype you want to use. Click OK to return to the Layer Properties Manager.
5 Click Apply to save your changes, or click OK to save and close.

Toolbar: Layers
Command entry: LAYER

To rename a layer

1 Click Home tab ➤ Layers panel ➤ Layer Properties
2 In the Layer Properties Manager, select a layer. Click the name or press F2.
3 Enter a new name.
4 Click Apply to save your changes, or click OK to save and close.

Toolbar: Layers
Command entry: LAYER

To rename more than one layer

1 Click Tools tab ➤ Drawing Utilities panel ➤ Rename.
2 In the Rename dialog box, in the Named Objects list, select Layers.
3 In Old Name, enter the old name, using wild-card characters; for example, enter **stairs$***.

4 In Rename To, enter the new name using wild-card characters; for example, enter **s_***.

   Results for this example are as follows: the layers STAIR$LEVEL-1, STAIR$LEVEL-2, STAIR$LEVEL-3 are renamed S_LEVEL-1, S_LEVEL-2, S_LEVEL-3.

5 Click Rename To to apply changes and continue, or click OK.

Command entry: RENAME

To undo changes to layer settings

- Click Home tab ➤ Layers panel ➤ Previous. 

The Command prompt displays the message “Restored previous layer status.”

NOTE LAYERP does not restore deleted or purged layers and does not remove a layer that was added. If you rename a layer and change its properties, Layer Previous restores the original properties but not the original layer name.

Toolbar: Layers 
Command entry: LAYERP

To set Layer Previous tracking on or off

1 At the Command prompt, enter **layerpmode**.

   The current Layer Previous tracking status is displayed.

2 Enter **on** to turn on Layer Previous tracking of layer settings, or enter **off** to turn off tracking.

Command entry: LAYERPMODE

Quick Reference

Commands

CHANGE

Changes the properties of existing objects.

CHPROP

Changes the properties of an object.

CLASSICLAYER

Opens the modal Layer Properties Manager.

LAYER

Manages layers and layer properties.

LAYERP

Undoes the last change or set of changes made to layer settings.

LAYERPALETTE

Opens the modeless Layer Properties Manager.

LAYERPMODE

Turns the tracking of changes made to layer settings on and off.

PROPERTIES

Controls properties of existing objects.

PURGE

Removes unused items, such as block definitions and layers, from the drawing.

RENAME

Changes the names assigned to items such as layers and dimension styles.

SETBYLAYER

Changes the property overrides of selected objects to ByLayer.

System Variables

CLAYER

Sets the current layer.

LAYERDLGMODE

Sets the flavor of the Layer Properties Manager that is defined for the use of the LAYER command.

LAYERMANAGERSTATE

Indicates whether the Layer Properties Manager is open or closed.

SETBYLAYERMODE

Controls which properties are selected for the SETBYLAYER command.

Utilities

No entries

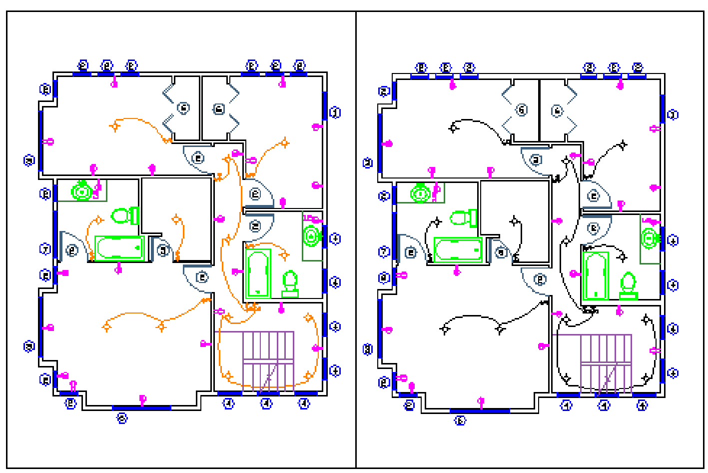
Command Modifiers

No entries

Override Layer Properties in Viewports

You can display objects differently by setting property overrides for color, linetype, lineweight, and plot style and apply them to individual layout viewports.

Using property overrides is an efficient way for displaying objects with different property settings in individual viewports without changing their ByLayer or ByBlock properties. For example, objects can be made to display more prominently by changing their color. Because layer property overrides do not change the layer's global properties, you can have objects display differently in various viewports without having to create duplicate geometry or use xrefs that have different layer settings.

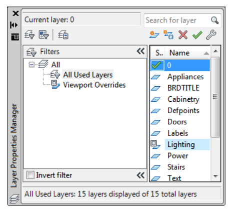


Property override settings for color and lineweight were set on the Wiring layer for the viewport on the left. Notice the wiring is a different color and lineweight than in the right viewport.

When the Layer Properties Manager is accessed from a layout tab, four columns for layer property overrides are displayed

- VP Color
- VP Lineweight
- VP Linetype
- VP Plot Style (available only in named-plot style drawings)

When a property override is set for a layer, a Viewport Overrides filter is automatically created in the Layer Properties Manager.



If you do not want to display or plot property overrides, set the VPLAYEROVERRIDESMODE system variable to 0. Objects will display and plot with their global layer properties.

NOTE Property overrides can still be set even when VPLAYEROVERRIDESMODE is set to 0.

Property overrides that are on xref layers are not retained when the VISRETAIN system variable is set to 0.

Viewport Property Overrides and Visual Styles

Layer property overrides for color, linetype, and lineweights are displayed in viewports regardless of the visual style that is current. Although plot style overrides can be set when the visual style is set to Conceptual or Realistic style, they are not displayed or plotted.

Identify Layers with Property Overrides

Layers containing property overrides are identifiable in the Layer Properties Manager when accessed from a layout tab. You can see which layers have overrides by the following:

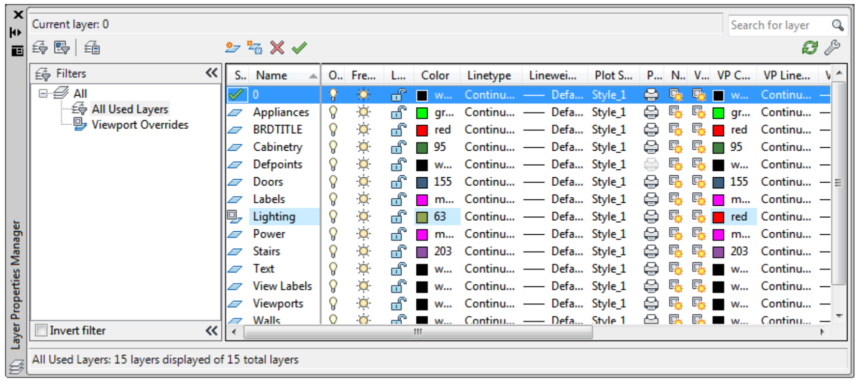
- A background color displays for each layer name, override and corresponding global property setting.

- A tooltip displays property override information when the cursor is placed over the status icon for the layer containing overrides.
- A different icon displays in the Status column.
- A predefined filter named Viewport Overrides is displayed in the tree view where all layers with viewport overrides are listed.

Layers that are assigned property overrides are also identifiable outside of the Layer Properties Manager. Other areas of the user interface that indicate which layers are assigned property overrides are as follows:

- Layers toolbar. A background color displays behind layer names for the current viewport. For color overrides, the override color is displayed instead of the global color.
- Properties toolbar. Displays ByLayer (VP) and a background color for layers or objects that are assigned property overrides. Override properties are displayed for color, linetype, and lineweight instead of global properties.
- Layers panel on the ribbon. A background color displays behind the layer names in the Layer drop-down list. Override properties are displayed for color, linetype, and lineweight instead of global properties.
- Properties panel on the ribbon. A background color displays behind the color, linetype, lineweight, and plot style controls. Override properties are displayed for color, linetype, and lineweight instead of global properties.
- Properties palette. Displays ByLayer (VP) and a background color for layers or objects that are assigned property overrides. Override properties are displayed for color, linetype, and lineweight instead of global properties.

NOTE The background color for viewport overrides can be changed in the Layer Settings dialog box.



Property overrides that are set on the Lighting layer for the current viewport are indicated by a blue background color.

Identify Viewports with Layer Property Overrides

Use the VPLAYEROVERRIDES system variable to check if the current viewport contains layer property overrides. When VPLAYEROVERRIDES is equal to 1, the viewport contains overrides.

You can also use the Properties palette to determine if a viewport contains overrides. The Properties palette displays a Layer Property Overrides field. The value that displays is the same as the setting for VPLAYEROVERRIDES.

Remove Layer Property Overrides

When you right-click a layer in the Layer Properties Manager, a shortcut menu is displayed that lists options for removing property overrides. You can remove

- A single property override from the selected layer for the selected viewport or for all selected viewports
- All property overrides from the selected layer for the selected viewport or for all selected viewports
- All property overrides from all layers in the selected viewport or for all selected viewports

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NOTE Another method for removing property overrides is to use the shortcut menu when you right-click the border of the selected viewport or viewports. You can remove viewport overrides for all layers for that viewport.

To assign property overrides for the current layout viewport

1 On the layout tab, double-click inside a viewport to make it current.

2 Click Home tab ➤ Layers panel ➤ Layer Properties

3 In the Layer Properties Manager, select the global properties to override in the VP Color, VP Linetype, VP Lineweight, and VP Plot Style columns.

Toolbar: Layers
Command entry: LAYER

To remove an override from a layer for the current layout viewport

1 On the layout tab, double-click inside a viewport to make it current.

2 Click Home tab ➤ Layers panel ➤ Layer Properties

3 In the Layer Properties Manager, right-click the property override on the layer you want to remove.

4 Click Remove Viewport Overrides For ➤ Property Override (Color, Linetype, Lineweight, Plot Style) ➤ In Current Viewport Only.

Toolbar: Layers
Command entry: LAYER

To remove an override from a layer for all layout viewports

1 On the layout tab, double-click inside a viewport to make it current.

2 Click Home tab ➤ Layers panel ➤ Layer Properties

3 In the Layer Properties Manager, right-click the property override on the layer to remove.

4 Click Remove Viewport Overrides For ➤ Property Override (Color, Linetype, Lineweight, Plot Style) ➤ In All Viewports.

Toolbar: Layers
Command entry: LAYER

To remove all overrides from a layer for the current layout viewport

1 On the layout tab, double-click inside a viewport to make it current.

2 Click Home tab ➤ Layers panel ➤ Layer Properties

3 In the Layer Properties Manager, right-click the layer.

4 Click Remove Viewport Overrides For ➤ Selected Layers ➤ In Current Viewport Only.

Toolbar: Layers
Command entry: LAYER

To remove all overrides for all layout viewports

1 Click a layout tab.

2 Click Home tab ➤ Layers panel ➤ Layer Properties

3 In the Layer Properties Manager, select the Viewport Overrides filter.

4 Right-click on any layer. Click Remove Viewport Overrides For ➤ All Layers ➤ In All Viewports.

Toolbar: Layers
Command entry: LAYER

To check if the current layout viewport contains layer property overrides

1 Double-click within a viewport to make it current.

2 At the Command prompt, enter **vplayeroverrides**.

If VPLAYEROVERRIDES displays 1, the selected viewport contains layer viewport overrides. If 0 is displayed, no overrides are found.

Command entry: VPLAYEROVERRIDES

To not display or plot layer viewport overrides

1 At the Command prompt, enter vplayeroverridesmode.

2 Enter **0**.

Command entry: VPLAYEROVERRIDESMODE

To change the background color for property overrides

1 Click Home tab ➤ Layers panel ➤ Layer Properties 

2 In the Layer Properties Manager, click Settings.

3 In the Layer Settings dialog box, select a color for the viewport override background color.

4 Click OK.

Toolbar: Layers 
Command entry: LAYER

To save layer viewport overrides in a layer state

1 On a layout tab, double-click in a viewport to make it active.

2 Click Home tab ➤ Layers panel ➤ Layer States Manager

3 In the Layer States Manager, click New.

4 In the New Layer State to Save dialog box, enter a name for the new layer state, or select a name from the list. (Optional) Add a description.

5 Click OK.

6 Click Close to exit the Layer State Manager.

Toolbar: Layers 

Command entry: LAYERSTATE

Quick Reference

Commands

CHPROP

Changes the properties of an object.

CLASSICLAYER

Opens the modal Layer Properties Manager.

LAYER

Manages layers and layer properties.

LAYERP

Undoes the last change or set of changes made to layer settings.

LAYERPALETTE

Opens the modeless Layer Properties Manager.

LAYERPMODE

Turns the tracking of changes made to layer settings on and off.

PROPERTIES

Controls properties of existing objects.

PURGE

Removes unused items, such as block definitions and layers, from the drawing.

RENAME

Changes the names assigned to items such as layers and dimension styles.

System Variables

CLAYER

Sets the current layer.

LAYERDLGMODE

Sets the flavor of the Layer Properties Manager that is defined for the use of the LAYER command.

VPLAYEROVERRIDES

Indicates if there are any layers with viewport (VP) property overrides for the current layout viewport.

VPLAYEROVERRIDESMODE

Controls whether layer property overrides for layout viewports are displayed and plotted.

Utilities

No entries

Command Modifiers

No entries

Filter and Sort the List of Layers

You can control which layer names are listed in the Layer Properties Manager and sort them by name or by property, such as color or visibility.

A layer filter limits the display of layer names in the Layer Properties Manager and in the Layer control on the Layers toolbar. In a large drawing, you can use layer filters to display only the layers you need to work with.

There are two kinds of layer filters

- **Layer property filter** Includes layers that have names or other properties in common. For example, you can define a filter that includes all layers that are red and whose names include the letters *mech.*
- **Layer group filter** Includes the layers that are put into the filter when you define it, regardless of their names or properties. Selected layers can be added from the layer list by dragging them to the filter.

The tree view in the Layer Properties Manager displays default layer filters and any named filters that you create and save in the current drawing. The icon next to a layer filter indicates the type of filter. Five default filters are displayed

- **All.** Displays all the layers in the current drawing. (Filter is always displayed.)
- **All Used.** Displays all the layers on which objects in the current drawing are drawn. (Filter is always displayed.)

- **Xref.** If xrefs are attached to the drawing, displays all the layers being referenced from other drawings.
- **Viewport Overrides**. If there are layers with overrides for the current viewport, displays all layers containing property overrides.
- **Unreconciled New Layers**. If new layers were added since the drawing was last opened, saved, reloaded, or plotted, displays a list of new unreconciled layers. See Reconcile New Layers on page 578 for more information.

NOTE The default filters cannot be renamed, edited, or deleted.

Once you have named and defined a layer filter, you can select it in the tree view to display the layers in the list view. You can also apply the filter to the Layers toolbar, so that the Layer control displays only the layers in the current filter.

When you select a filter in the tree view and right-click, options on the shortcut menu can be used to delete, rename, or modify filters. For example, you can convert a layer property filter to a layer group filter. You can also change a property of all layers in a filter. The Isolate Group option turns off all layers in the drawing that are not in the selected filter.

Define a Layer Property Filter

A layer property filter is defined in the Layer Filter Properties dialog box, where you select any of the following properties you want to include in the filter definition:

- Layer names, colors, linetypes, lineweights, and plot styles
- Whether layers are in use
- Whether layers are turned on or off
- Whether layers are frozen or thawed in the active viewport or all viewports
- Whether layers are locked or unlocked
- Whether layers are set to be plotted

You use wild-card characters to filter layers by name. For example, if you want to display only layers that start with the letters *mech*, you can enter **mech***. See "Wild-Card Characters" for a complete list.

The layers in a layer property filter may change as the properties of the layers change. For example, if you define a layer property filter named Site that includes all layers with the letters *site* in the name and a CONTINUOUS linetype, and then you change the linetype of some of those layers, the layers with the new linetype are no longer part of the Site filter and are not displayed when you apply that filter.

Layer property filters can be nested under other properties filters or under group filters.

Define a Layer Group Filter

A layer group filter includes only those layers that you explicitly assign to it. If the properties of the layers assigned to the filter change, the layers are still part of the filter. Layer group filters can be nested only under other layer group filters.

TIP Layers from the layer list can be included in a filter by clicking and dragging the selected layers to the filter.

Invert a Layer Filter

You can also invert a layer filter. For example, if all the site plan information in a drawing is contained in multiple layers that include the word *site* as part of the layer name, you can display all information except site plan information by first creating a filter definition that filters layers by name (**site**) and then using the Invert Filter option.

Sort Layers

Once you have created layers, you can sort them by name or other properties. In the Layer Properties Manager, click the column heading to sort layers by the property in that column. Layer names can be sorted in ascending or descending alphabetical order.

Wild-Card Characters

You can use wild-card characters to sort layers by name.

Character	Definition
# (pound)	Matches any numeric digit
@ (at)	Matches any alphabetic character
. (period)	Matches any nonalphanumeric character

Character	Definition
* (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
~ (tilde)	Matches anything but the pattern; for example; ~*AB*matches all strings that don't contain AB
[ ]	Matches any one of the characters enclosed; for example, [AB]C matches AC and BC
[~]	Matches any character not enclosed; for example, [~AB]C matches XC but not AC
[-]	Specifies a range for a single character; for example, [A-G]C matches AC, BC, and so on to GC, but not HC
` (reverse quote)	Reads the next character literally; for example, `~AB matches ~AB

NOTE To filter on a layer name that contains a wild-card character, precede the character with a reverse quote (`) so that it is not interpreted as a wild-card character.

See also:

- Reconcile New Layers on page 578

To quickly filter the display of layers by name

1. Click Home tab ➤ Layers panel ➤ Layer Properties 
2. In the Layer Properties Manager, click in Search for Layer below the tree view.
3. (Optional) To limit your search, select a layer filter in the tree view.
4. Enter a string of characters, including wild-card characters.

The list view displays all layers whose names match the string. For example, if you enter ***mech***, all layers with the letters *mech* in their names are displayed.

This quick filter is discarded when the Layer Properties Manager closes.

Toolbar: Layers 
Command entry: LAYER

To filter the display of layers by layer property

1 Click Home tab ➤ Layers panel ➤ Layer Properties 

2 In the Layer Properties Manager, click the New Property Filter button.

3 In the Layer Filter Properties dialog box, enter a name for the filter.

4 Under Filter Definition, set the layer properties that you want to use to define the filter.

  - To filter by name, use wild-card characters.
  - To filter by property, click in the column for the property you want. Some properties display a dialog box when you click the [...] button.
  - To select more than one value for a property, right-click the row in the filter definition. Click Duplicate Row. Select another value for that property in the next row.
    For example, the definition for a filter that displays only layers that are on and are either yellow or red has two rows. The first row of the filter definition has the On icon and red. The second row has the On icon and yellow.

5 Click Apply to save your changes, or click OK to save and close.

Toolbar: Layers 
Command entry: LAYER

To filter the display of layer names by selecting layers

1 Click Home tab ➤ Layers panel ➤ Layer Properties 

2 In the Layer Properties Manager, click the New Group Filter button.

   A new layer group filter named GROUP FILTER1 is created in the tree view.

3 Enter a name for the filter.

4 In the tree view, click All or one of the other nodes to display layers in the list view.

5 In the list view, select the layers you want to add to the filter, and drag them to the filter name in the tree view.

6 Click Apply to save your changes, or click OK to save and close.

Toolbar: Layers 
Command entry: LAYER

To nest a layer filter under another layer filter

1 Click Home tab ➤ Layers panel ➤ Layer Properties 

2 In the Layer Properties Manager tree view, select a layer filter.

   - A new layer property filter can be nested under a group filter or another property filter.
   - A new layer group filter can be nested only under another group filter.

3 Right-click. Click New Properties Filter or New Group Filter.

4 Use one of the following methods:

   - For a new property filter, the Layer Properties Filter dialog box is displayed. Under Filter Definition, set the layer properties that you want to use to define the filter. Click OK.
   - For a new group filter, a filter is added to the tree view. Rename it, select the parent filter to display its layers in the list view, and drag layers from the list view to the new layer group filter.

5 Click Apply to save your changes, or click OK to save and close.

Toolbar: Layers
Command entry: LAYER

To sort the layer list in the Layer Properties Manager

- Click Home tab ➤ Layers panel ➤ Layer Properties
- In the Layer Properties Manager, click any column heading.

To reverse the sorting order, click a second time.

Toolbar: Layers
Command entry: LAYER

To remove a layer from a layer group filter

1 Click Home tab ➤ Layers panel ➤ Layer Properties
2 In the Layer Properties Manager tree view, select a group filter.
3 Select the layer you want to remove.
4 Right-click in the list view. Click Remove from Group Filter.

Toolbar: Layers
Command entry: LAYER

Quick Reference

Commands

CLASSICLAYER

Opens the modal Layer Properties Manager.

LAYER

Manages layers and layer properties.

LAYERPALETTE

Opens the modeless Layer Properties Manager.

System Variables

LAYERDLGMODE

Sets the flavor of the Layer Properties Manager that is defined for the use of the LAYER command.

MAXSORT

Sets the maximum number of symbol names or block names sorted by listing commands.

Utilities

No entries

Command Modifiers

No entries

Use New Layer Notification

You can be notified when new layers are added to the drawing before certain tasks, such as plotting, saving, or restoring a layer state.

It is important to be aware of new layers that have been added to a drawing or to an attached xref without your knowledge. You can avoid potential problems, such as plotting objects that were added to the drawing by the addition of a new layer.

You can control when to evaluate a drawing for new layers. You can specify which commands, such as SAVE or PLOT, trigger the program to check the layer list and alert you of new layers. This can include new layers that have been added to attached xrefs.

The LAYEREVAL and LAYERNOTIFY system variables work together to control whether the layer list is evaluated and when notification occurs. Both system variables are saved in the drawing so you have control over which drawings are checked for new layers. When a project is started, it may not be necessary to know when new layers have been created. For drawings that are nearing completion, it may be important to be aware if new information has been introduced into the drawing from the addition of new layers.

The LAYEREVALCTL controls the overall Unreconciled New Layer filter list in the Layer Properties Manager which is evaluated for new layers. When the New Layer Notification is checked (LAYEREVALCTL = 1), the new layer notification feature is enabled and functions based on the LAYEREVAL and LAYERNOTIFY drawing system variables. There should be no filter (Unreconciled Layer filter) displayed when LAYEREVALCTL = 0. If one is currently displayed, it will be turned off.

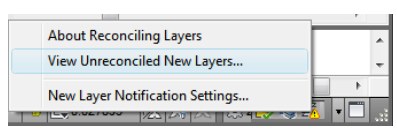
By default, LAYEREVAL is set to detect any new layers that have been added to attached xrefs but not in the drawing. LAYERNOTIFY is set to notify you of new layers when opening the drawing, when loading, reloading, or attaching xrefs, or when restoring a layer state. To make changes to these settings, you can either use the system variables or the Layer Settings dialog box.

When layer notification is turned on, an Unreconciled New Layers icon displays on the status bar.

At that time, you can choose to view the new layers by right-clicking the icon and clicking the View Unreconciled New Layers link from the menu. When you click the link, the Layer Properties Manager opens, and the Unreconciled New Layers filter is automatically selected. All new layers that have been added to the drawing or attached xrefs are displayed in the list view.

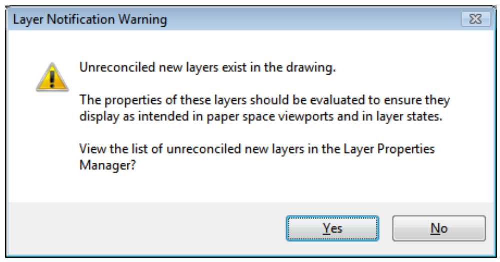
The new layers are *unreconciled* because they have not yet been reviewed. The process of manually marking them as *reconciled* removes them from the Unreconciled New Layers list. (See Reconcile New Layers on page 578 for more information.) Until you reconcile the layers, the notification bubble will display each time the command that triggered the notification is used.

By using the LAYERNOTIFY system variable or the Layer Settings dialog box, you can turn off layer notification but still have the layer list evaluated for new layers. Although the notification bubble does not display, you can still check for new layers by right-clicking the alert icon on the status bar. From the shortcut menu, click the View Unreconciled New Layers option.



New Layer Notification for Plotting

When the PLOT command is set to display new layer notification in the Layer Settings dialog box, a dialog box is displayed instead of an icon on the status bar. A message informs you that there are new layers in the drawing since the layer list was last checked for new layers. Click Yes in the dialog box to view the new layer list in the Layer Properties Manager before plotting.



Saving a Template (DWT) File

When saving a drawing as a template (dwt) file, you can choose to save the layers in the drawing as unreconciled or reconciled in the Template Options dialog box. By default, all layers are saved as unreconciled, so that when a new drawing is started using the template, a layer baseline is not yet created until the drawing is first saved as a *.dwg* file.

If the template file is saved with all layers as reconciled, a layer baseline is created. That means when new layers are added to the drawing that is created from the template file, any new layers that are created are unreconciled and a new layer notification will display when the drawing is first saved or plotted.

Opening Multiple Drawings

When opening multiple drawings at the same time, an alert displays for each drawing that contains new layers. This behavior occurs if layer notification is turned on and the OPEN command is specified in the LAYERNOTIFY system variable for each drawing.

To set new layer notification on and off

1. Click Home tab ➤ Layers panel ➤ Layer Properties 
2. In the Layer Properties Manager, click Settings.
3. In the Layer Settings dialog box, click Evaluate New Layers Added to Drawing.

4 Click Notify When New Layers are Present. Select one or more options.

5 Click OK.

6 Click OK to exit the Layer Properties Manager.

Toolbar: Layers
Command entry: LAYER

To set when new layer notification is displayed

1 Click Home tab ➤ Layers panel ➤ Layer Properties

2 In the Layer Properties Manager, click Settings.

3 In the Layer Settings dialog box, click Evaluate New Layers Added to Drawing.

4 Click Notify When New Layers are Present.

5 Select the commands that will cause the layer list to be evaluated for new layers.

6 Click OK.

Toolbar: Layers
Command entry: LAYER

Quick Reference

Commands

CLASSICLAYER

Opens the modal Layer Properties Manager.

LAYER

Manages layers and layer properties.

LAYERPALETTE

Opens the modeless Layer Properties Manager.

System Variables

LAYERDLGMODE

Sets the flavor of the Layer Properties Manager that is defined for the use of the LAYER command.

LAYEREVALCTL

Controls the overall Unreconciled New Layer filter list in Layer Properties Manager which is evaluated for new layers.

LAYERNOTIFY

Specifies when an alert displays when unreconciled new layers are found.

LAYEREVAL

Specifies whether the layer list is evaluated for new layers when added to the drawing or to attached xrefs.

Utilities

No entries

Command Modifiers

No entries

Reconcile New Layers

Unreconciled layers are new layers that have been added to the drawing and have not yet been acknowledged by the user and manually marked as reconciled.

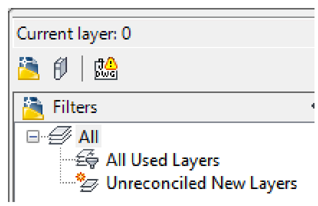
Reconciling new layers is the process of manually reviewing new layers so that you can avoid potential errors before plotting your drawing or when restoring a layer state.

Unreconciled layers are new layers that have been added to the drawing or to attached xrefs since the layer list was last evaluated. The layer list is checked for new layers when a command, such as PLOT is used. In new drawings, the layer baseline is created when the drawing is saved or plotted for the first time. When a new drawing is first saved, the layer baseline is created, and all layers present in the saved drawing are considered reconciled (not new). Layers that are added after a drawing is first saved are considered new unreconciled layers.

NOTE The layer baseline is created when the LAYEREVAL system variable is set to 1 or 2.

When a command that is set in the Layer Settings dialog box or LAYEREVAL system variable is used, the layer list is checked at that time and compared to the baseline. If there are new layers, notification will display and the Unreconciled New Layers filter is automatically created and activated in the Layer Properties Manager.

Unreconciled layers become reconciled by right-clicking the layer and clicking the Reconcile Layer option. Once a layer has become reconciled, it is removed from the Unreconciled New Layers filter. After all new layers are reconciled, the Unreconciled New Layers filter is removed.



NOTE You can reconcile multiple unreconciled layers at the same time.

Quick Reference

Commands

CLASSICLAYER

Opens the modal Layer Properties Manager.

LAYER

Manages layers and layer properties.

LAYERPALETTE

Opens the modeless Layer Properties Manager.

System Variables

LAYERDLGMODE

Sets the flavor of the Layer Properties Manager that is defined for the use of the LAYER command.

LAYERNOTIFY

Specifies when an alert displays when unreconciled new layers are found.

LAYEREVAL

Specifies whether the layer list is evaluated for new layers when added to the drawing or to attached xrefs.

Utilities

No entries

Command Modifiers

No entries

Work with Layer States

You can save layer settings as named layer states. You can then restore, edit, import them from other drawings and files, and export them for use in other drawings.

Save, Restore, and Edit Layer States

You can save the current layer settings to a layer state, make changes to the layer state, and restore them to the drawing later.

You can save the current layer settings in a drawing as a named layer state and restore them later. Saving layer settings is convenient if you need to return to particular settings for all layers during different stages in completing a drawing or for plotting.

Save Layer Settings

Layer settings include layer states, such as on or locked, and layer properties, such as color or linetype. In a named layer state, you can choose which layer states and layer properties you want to restore later. For example, you can choose to restore only the Frozen/Thawed setting of the layers in a drawing, ignoring all other settings. When you restore that named layer state, all settings remain as they are currently set except whether each layer is frozen or thawed.

Save Layer Property Override Settings

When layers contain viewport property overrides, those settings are saved to a layer state when the viewport that contains overrides is active.

If the layer state is saved from model space, any layer property override settings are not included. This is because only one value can be saved for each layer property in a layer state. If layer property overrides need to be saved in the layer state, make the viewport active on the layout tab and then save the layer state.

Restore Layer Settings

When you restore a layer state, the layer settings (layer states and layer properties) that were specified when the layer state was saved are restored. You can specify specific settings to restore in the Layer States Manager. The layer property settings that are not selected remain unchanged in the drawing.

If the drawing contains layers that were added since a layer state was saved, you can add those layers. By editing the layer state and using the Select Layers to Add to Layer State dialog box you can select the layers you want to add to the layer state.

NOTE To be notified when new layers are added to the drawing, use the LAYEREVAL and LAYERNOTIFY system variables.

When restoring layer states, the following additional behaviors can occur

- When restoring a layer state, the layer that was current when the layer state was saved is made current. If that layer no longer exists, the current layer does not change.
- If a layout viewport is active when a layer state is restored, and the Visibility in Current VP restore option is selected, all layers that need to be visible in the viewport are turned on and thawed in model space. All layers that should not be visible in the viewport are set to VP Freeze in the current viewport and the model space visibility is unchanged.

The Layers panel on the RIBBON contains controls for selecting and restoring saved layer states, including a button to access the Layer States Manager. The name of the last restored layer state is displayed in the Layers panel. When the layers are modified so that the current layer state is no longer current, “Unsaved Layer State” is displayed instead of the name of the last restored layer state.

Restore Property Override Settings

When the Apply Properties as Viewport Overrides restore option is selected in the Layer States Manager, viewport overrides are restored to the viewport that is current at the time the layer state is restored.

When a layer state is saved in model space and is restored in paper space,

- You can choose whether color, linetype, lineweight, or plot style properties are restored as viewport overrides.
- Viewport overrides are applied to the current layout viewport.
- Layers that were turned off or frozen in model space are set to VP Freeze in the Layer Properties Manager for the active layout viewport.

When a layer state is saved in paper space and is restored in model space,

- Layer property overrides are restored as global layer properties in model space.
- Layers that were frozen in a layout viewport are also frozen in model space.

Edit Saved Layer Settings

Using the Edit Layer State dialog box, you can modify the properties of each layer saved in a layer state.

All properties other than the layer name can be edited. Properties for multiple layers can be changed at the same time.

You can also add layers to a layer state through the Select Layers to Add to Layer State dialog box. For example, if new layers were added since the layer state was saved, you can add them and resave the layer state. To delete layers, use the Delete button in the Edit Layer State dialog box.

Layer States in Xrefs

When a drawing containing layer states is inserted in the host drawing, the xref layer states are displayed in the Layer States Manager. The layer states are listed by name and can be viewed in the Edit Layer State dialog box.

When an xref containing layer states is attached to the host drawing, those layer states are also listed in the Layer States Manager. Although they can be restored, they cannot be edited. Xref layer states are identifiable because the layer state name is preceded by the xref drawing's name and separated by a double underscore symbol. (Example: *Xref Name__Layer State Name*.) When the xref is bound to the host drawing, layer states are identifiable by 0 that displays between the xref name and layer state name. (Example: *Xref Name0Layer State Name*.)

Layer states from nested xrefs are also included. Layer states from xrefs are removed from the host drawing when the xref is detached or unloaded.

To save layer settings in a named layer state

1 Click Home tab ➤ Layers panel ➤ Layer State.

2 In the Layer States drop-down list, click New Layer State.

3 In the New Layer State to Save dialog box, enter a name for the new layer state, or select a name from the list.

4 (Optional) Add a description.

5 Click Close.

6 In the Layer States Manager, select the layer properties to restore by default.

7 (Optional) Select the Turn Off New Layers Not Found in Layer State option.

   When this option is selected and you restore a named layer state, the drawing looks the same way it did when the named layer state was saved.

8 Click Close to exit the Layer States Manager.

Toolbar: Layer States Manager 
Command entry: LAYERSTATE

To restore a layer state

1 Click Home tab ➤ Layers panel ➤ Layer State.

2 In the Layer State drop-down list, select Manage Layer States.

3 In the Layer States Manager dialog box, select a named layer state.

4 Click More to select any specific layer properties you want to restore.

5 Click Restore.

   The Layer States Manager closes.

Toolbar: Layer States Manager 
Command entry: LAYERSTATE

To add layers to a layer state

1 Click Home tab ➤ Layers panel ➤ Layer State.

2 In the Layer State drop-down list, select Manage Layer States.

3 In the Layer States Manager dialog box, select the named layer state that you want to add layers to.

4 Click Edit.

5 In the Edit Layer State dialog box, click Add.

6 In the Select Layers to Add to Layer State dialog box, select the layers you want to add.

7 Click OK.

8 Click OK to exit the Edit Layer State dialog box.

9 Click Close to exit the Layer States Manager.

Toolbar: Layer States Manager 
Command entry: LAYERSTATE

To delete layers from a layer state

1 Click Home tab ➤ Layers panel ➤ Layer State.

2 In the Layer State drop-down list, select Manage Layer States.

3 In the Layer States Manager dialog box, select the named layer state you want to delete layers from.

4 Click Edit.

5 In the Edit Layer State dialog box, select the layers to delete and click the Delete button.

6 Click OK.

7 Click OK to exit the Edit Layer State dialog box.

8 Click Close to exit the Layer States Manager.

Toolbar: Layer States Manager 

Command entry: LAYERSTATE

To modify a named layer state

1 Click Home tab ➤ Layers panel ➤ Layer State.

2 In the Layer State drop-down list, select Manage Layer States.

3 In the Layer States Manager dialog box, select the named layer state you want to modify.

4 Click Edit.

5 In the Edit Layer State dialog box, select the property cells to change.

6 Click OK.

7 Click Close to exit the Layer States Manager.

Toolbar: Layer States Manager 
Command entry: LAYERSTATE

To rename a named layer state

1 Click Home tab ➤ Layers panel ➤ Layer State.

2 In the Layer State drop-down list, select Manage Layer States.

3 In the Layer States Manager dialog box, select the named layer state you want to rename.

4 Click Rename.

5 In the Name field, enter a new name.

6 Click Close to exit the Layer States Manager.

NOTE An alternate method is to select the layer state name and press F2 for in-place editing.

Toolbar: Layer States Manager 
Command entry: LAYERSTATE

To delete a named layer state

1 Click Home tab ➤ Layers panel ➤ Layer State.

2 In the Layer State drop-down list, select Manage Layer States.

3 In the Layer States Manager dialog box, select the named layer state to delete.

4 Click Delete.

5 Click Close to exit the Layer States Manager.

Toolbar: Layer States Manager
Command entry: LAYERSTATE

To include description and material properties to a layer state imported from a previous release

1 Click Home tab ➤ Layers panel ➤ Layer State.

2 In the Layer State drop-down list, select Manage Layer States.

3 In the Layer States Manager, select the layer state that was imported from a previous release.

4 Click Restore to restore the layer state.

5 Open the Layer States Manager. Click New.

6 In the New Layer State to Save dialog box, enter a new name for the layer state. In the Description field, enter descriptive text about the layer settings.

7 Click OK.

8 (Optional) Click Delete to remove the legacy layer state.

9 Click Close.

Description and material properties are saved with the updated layer state.

Toolbar: Layer States Manager
Command entry: LAYERSTATE

To select a layer state to restore

1 Click Home tab ➤ Layers panel ➤ Layer State.

2 In the Layer State drop-down list, select Manage Layer States.

3 In the Layer States Manager dialog box, select the layer state you want to restore.

4 Click More and select any specific layer properties you want to restore.

5 Click Restore.

The Layer States Manager closes.

Toolbar: Layer States Manager 
Command entry: LAYERSTATE

To restore layer viewport overrides

1 Select a layout viewport.

2 Click Home tab ➤ Layers panel ➤ Layer State.

3 In the Layer State drop-down list, select Manage Layer States.

4 In the Layer States Manager dialog box, select the named layer state to restore.

5 Select Restore as Viewport Overrides.

6 Click Restore.

The Layer States Manager closes. The saved values for color, linetype, lineweight, and plot style are applied to the layers as overrides in the current viewport.

Toolbar: Layer States Manager 
Command entry: LAYERSTATE

Quick Reference

Commands

LAYERSTATE

Saves, restores, and manages named layer states.

System Variables

LAYERNOTIFY

Specifies when an alert displays when unreconciled new layers are found.

LAYEREVAL

Specifies whether the layer list is evaluated for new layers when added to the drawing or to attached xrefs.

Utilities

No entries

Command Modifiers

No entries

Import and Export Layer States

You can import layer settings from other drawings and export layer states.

You can import layer states that are saved in drawing files (DWG, DWS, and DWT) and from layer state (LAS) files. When importing layer states from a drawing file, you can choose multiple layer states to import from the Select Layer States dialog box. When exporting layer states, they are created as LAS files.

If the layer state is imported from a drawing and it contains a layer property, such as a linetype or plot style that is not loaded or available in the current drawing, that property is automatically imported from the source drawing.

If the layer state is imported from an LAS file, and it contains linetype or plot style properties that do not exist in the drawing, a message is displayed notifying that the property could not be restored.

NOTE When a layer state contains more than one property that cannot be restored from an LAS file, the message that displays only indicates the first property it encountered that cannot be restored.

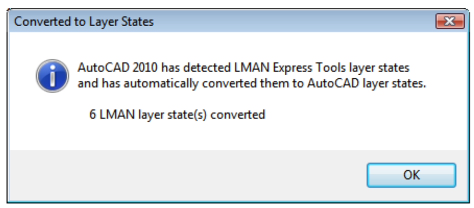
When importing a layer state from an LAS file or from another drawing that are duplicates of layer states in the current drawing, you can choose to overwrite the existing layer state or not import it.

Layer states can be imported into a previous release of the program.

Layer States from LMAN

Layer states that were created using the LMAN Express Tool cannot be imported. A message is displayed that there are no layer states to import.

You can access LMAN layer states in a drawing through the Layer States Manager. When the Layer States Manager is first opened in a drawing containing LMAN layer states, they are automatically converted to AutoCAD layer states. A dialog box displays the number of layer states that have been converted.



When the current drawing does not contain any named layer states, the LMAN layer state names are retained. If the current drawing contains layer states, LMAN layer state names display with the prefix “LMAN” followed by the original layer state name.

To import saved layer settings from another drawing

1. Click Home tab ➤ Layers panel ➤ Layer State.
2. In the Layer State drop-down list, select Manage Layer States.
3. In the Layer States Manager dialog box, click Import.
4. In the Import Layer State dialog box, select a file name with a *.dwg, .dws, or .dwt* file name extension. Click Open.
5. In the Select Layer States dialog box, select the layer states to import. Click OK.

6 To restore the named layer state now, select it in the Layer States Manager. and click Restore. Click Close to not restore it.

   If you restore the named layer state, the Layer States Manager closes.

7 Click Close to exit the Layer States Manager.

Toolbar: Layer States Manager 
Command entry: LAYERSTATE

To import saved layer settings from a layer state (LAS) file

1 Click Home tab ➤ Layers panel ➤ Layer State.

2 In the Layer State drop-down list, select Manage Layer States.

3 In the Layer States Manager dialog box, click Import.

4 In the Import Layer State dialog box, select the LAS file you want to import layer states from. Click Open.

5 Click Yes to restore the named layer state now. Click No to add it to the Layer States Manager without restoring it.

   If you restore the named layer state, the Layer States Manager closes.

6 Click Close to exit the Layer States Manager.

Toolbar: Layer States Manager 
Command entry: LAYERSTATE

To export a saved layer state

1 Click Home tab ➤ Layers panel ➤ Layer State.

2 In the Layer State drop-down list, select Manage Layer States.

3 In the Layer States Manager dialog box, select the named layer state (LAS) file you want to export. Click Export.

4 In the Export Layer State dialog box, specify where to export the layer state file.

5 Click Save to exit the dialog box.

6 Click Close to exit the Layer States Manager.

Toolbar: Layer States Manager 
Command entry: LAYERSTATE

Quick Reference

Commands

LAYERSTATE

Saves, restores, and manages named layer states.

System Variables

LAYERNOTIFY

Specifies when an alert displays when unreconciled new layers are found.

LAYEREVAL

Specifies whether the layer list is evaluated for new layers when added to the drawing or to attached xrefs.

Utilities

No entries

Command Modifiers

No entries

Work with Colors

Color helps to group objects visually. You can assign colors to objects by layer or individually.

Set the Current Color

You can use color to help you identify objects visually. You can assign the color of an object either by layer or by specifying its color explicitly, independent of layer.

You can assign the color of an object either by layer or by specifying its color explicitly, independent of layer. Assigning colors by layer makes it easy to identify each layer within your drawing. Assigning colors explicitly provides additional distinctions between objects on the same layer. Color is also used as a way to indicate lineweight for color-dependent plotting.

You can use a variety of color palettes when assigning color to objects, including

- AutoCAD Color Index (ACI)
- True Color®, PANTONE®
- RAL™ Classic and RAL Design color books
- DIC® Color Guide
- Colors from imported color books.

ACI Colors

ACI colors are the standard colors used in AutoCAD. Each color is identified by an ACI number, an integer from 1 through 255. Standard color names are available only for colors 1 through 7. The colors are assigned as follows: 1 Red, 2 Yellow, 3 Green, 4 Cyan, 5 Blue, 6 Magenta, 7 White/Black.

True Colors

True colors use 24-bit color definitions to display over 16 million colors. When specifying true colors, you can use either an RGB or HSL color model. With the RGB color model, you can specify the red, green, and blue components of the color; with the HSL color model, you can specify the hue, saturation, and luminance aspects of the color.

Color Books

This program includes several standard Pantone color books. You can also import other color books such as the DIC color guide or RAL color sets. Importing user-defined color books can further expand your available color selections.

NOTE Pantone has provided new color definitions for Architectural & Interiors Cotton and Architectural & Interiors Paper color books. If you used these color books in releases prior to AutoCAD 2006, you may notice subtle changes in the colors.

You install color books on your system by using the Files tab in the Options dialog box. Once a color book is loaded, you can select colors from the color book and apply them to objects in your drawings.

All objects are created using the current color, which is displayed in the Color control on the Properties toolbar. You can also set the current color with the Color control or the Select Color dialog box.

If the current color is set to BYLAYER, objects are created with the color assigned to the current layer. If you do not want the current color to be the color assigned to the current layer, you can specify a different color.

If the current color is set to BYBLOCK, objects are created using color 7 (white or black) until the objects are grouped into a block. When the block is inserted into the drawing, it acquires the current color setting.

To set an ACI color for all new objects

1 Click Home tab ➤ Properties panel ➤ Object Color. 

2 In the Object Color drop-down list, click a color to draw all new objects in that color, or click Select Color to display the Select Color dialog box and do one of the following:
   - On the Index Color tab, click a color or enter the color name or number in the Color box.
   - On the Index Color tab, click BYLAYER to draw new objects in the color assigned to the current layer.
   - On the Index Color tab, click BYBLOCK to draw new objects in the current color until they are grouped into a block. When the block is inserted into the drawing, the objects in the block acquire the current color setting.

3 Click OK.

   The Color control displays the current color.

Command entry: COLOR

To set a true color for all new objects

1 Click Home tab ➤ Properties panel ➤ Object Color. 

2 In the Object Color drop-down list. click Select Color to display the Select Color dialog box.

3 In the Select Color dialog box, True Color tab, do one of the following:

- Select the HSL color model in the Color Model box. Specify a color by entering a color value in the Color box or by specifying values in the Hue, Saturation, and Luminance boxes.

- Select the RGB color model in the Color Model box. Specify a color by entering a color value in the Color box or by specifying values in the Red, Green, and Blue boxes.

4 Click OK.

The Color control displays the current color.

Command entry: COLOR

To set a color from a color book for all new objects

1 Click Home tab ➤ Properties panel ➤ Object Color. 

2 In the Object Color drop-down list, click Select Color.

3 In the Select Color dialog box, Color Book tab, select a color book from the Color Book box.

4 Select a color by clicking on a color chip. To browse through the color book, use the up and down arrows on the color slider.

5 Click OK.

The Color control displays the current color.

Command entry: COLOR

Quick Reference

Commands

COLOR

Sets the color for new objects.

System Variables

CECOLOR

Sets the color of new objects.

Utilities

No entries

Command Modifiers

No entries

Change the Color of an Object

You can change the color of an object by reassigning it to another layer, by changing the color of the layer the object is on, or by specifying a color for the object explicitly.

You have three choices for changing the color of an object:

- Reassign the object to another layer with a different color. If an object's color is set to BYLAYER, and you reassign the object to a different layer, it acquires its color from the new layer.
- Change the color assigned to the layer that the object is on. If an object's color is set to BYLAYER, it acquires the color of its layer. When you change the color assigned to a layer, all objects on that layer assigned the BYLAYER color are updated automatically.
- Specify a color for an object to override the layer's color. You can specify the color of each object explicitly. If you want to override the layer-determined color of an object with a different one, change an existing object's color from BYLAYER to a specific color, such as red.

If you want to set a specific color for all subsequently created objects, change the current color setting on the Properties toolbar from BYLAYER to a specific color.

See also:

- Override Layer Properties in Viewports on page 559

To change the layer of an object

1 Select the objects whose layer you want to change.

2 Click Home tab ➤ Layers panel ➤ Layer Properties.

3 In the Layers Properties Manager, click the Layer control.

4 Select the layer that you want to assign to the objects.

Toolbar: Layers

To change the color assigned to a layer

1 Click Home tab ➤ Layers panel ➤ Layer Properties.

2 In the Layer Properties Manager, click the color you want to change.

3 In the Select Color dialog box, do one of the following:
   - On the Index tab, click a color or enter the ACI color number (1-255) or name in the Color box. Click OK.
   - On the True Color tab, select the HSL color model in the Color Model option and specify a color by entering a color value in the Color box or by specifying values in the Hue, Saturation, and Luminance boxes. Click OK.
   - On the Color Books tab, select a color book from the Color Book box; select a color by navigating the color book (using the up and down arrows) and clicking on a color chip. Click OK.

4 Click OK to close each dialog box.

Toolbar: Layers
Command entry: LAYER

To change the color of objects, overriding the layer's color

1 Select the objects whose color you want to change.

2 Click View tab ➤ Properties panel ➤ Properties. 

Alternatively, you can right-click one of the objects and then, click Properties.

3 In the Properties palette, select Color.

An arrow is displayed in the right column.

4 Click the arrow and select a color from the list or click Select Color to display the Select Color dialog box. If you click Select Color, do one of the following:

- On the Index tab, click a color or enter the ACI color number (1-255) or name in the Color box.
- On the True Color tab, select the HSL color model in the Color Model option and specify a color by entering a color value in the Color box or by specifying values in the Hue, Saturation, and Luminance boxes. Click OK.
- On the True Color tab, select the RGB color model in the Color Model box. Specify a color by entering a color value in the Color box or by specifying values in the Red, Green, and Blue boxes.
- On the Color Books tab, select a color book from the Color Book box; select a color by navigating the color book (using the up and down arrows) and clicking on a color chip. Click OK.

Command entry: PROPERTIES

Quick Reference

Commands

CHANGE

Changes the properties of existing objects.

CHPROP

Changes the properties of an object.

COLOR

Sets the color for new objects.

PROPERTIES

Controls properties of existing objects.

System Variables

CECOLOR

Sets the color of new objects.

Utilities

No entries

Command Modifiers

No entries

Use Color Books

When assigning colors to objects, you can choose colors from color books that are loaded on your system. You can choose from a wide range of custom colors when using color books. Color books include third-party or user-defined files that contain named color swatches. These colors can be used to enhance presentation drawings as well as to optimize the variety of color used in your drawings. You can apply color book colors to objects in your drawings by using the Color Books tab in the Select Color dialog box.

Install Color Books

Color book files must contain an *.acb* file extension in order to be recognized by this program. To access color book colors from the Select Color dialog box, you must first copy your color book files to a specified color book location. On the Files tab of the Options dialog box, you can define the path where color book files are stored. Multiple locations can be defined for the color book path. These locations are saved in your user profile.

After loading a color book on your system, to access the new colors, you need to close the Select Color dialog box and then open it again. The new color book is displayed in the Color Book drop-down list on the Color Books tab. Once you have loaded a color book, you can apply any colors that are defined in the book to objects in your drawing.

Browse Color Books

Color books are organized alphabetically into pages that you can browse through. A page holds up to ten colors. If the color book you are browsing through is not organized into pages, the colors are arranged into pages, with each page containing up to seven colors.

To install a color book

1 Click Tools menu ➤ Options. 

2 In the Options dialog box, Files tab, click Color Book Locations.

3 Click Add to add a color book location.

4 Enter the new location in the blank path box.

5 Click OK.

Command entry: OPTIONS

To search for color swatches within a color book

1 Click Home tab ➤ Properties panel ➤ Object Color. 

2 In the Object Color drop-down list, click Select Color.

3 In the Select Color dialog box, Color Books tab, select a color book from the Color Book drop-down list.

   You must first select any color swatch in the color book in order to activate the Color edit box.

4 Under Color, enter the number of the color swatch you would like to locate and press TAB.

   The Color edit box and the New color chip display the requested color or the color that is the closest match.

5 Click OK to apply the color.

Command entry: COLOR

To change the default location of color book files

1 Click Tools menu ➤ Options.
2 In the Options dialog box, Files tab, double-click Color Book Locations.
3 Select the following default color book location:
C:\Program Files\[current AutoCAD release number]\support\color
4 Click the location name to edit the path.
5 Enter the new location in the path box.
6 Click OK.

Command entry: OPTIONS

To define multiple folders for the color book path

1 Click Tools menu ➤ Options.
2 In the Options dialog box, Files tab, click Color Book Locations.
3 Click Add to add a color book location.
4 Enter the new location in the blank path box.
5 Click OK.

Command entry: OPTIONS

Quick Reference

Commands

CHANGE

Changes the properties of existing objects.

CHPROP

Changes the properties of an object.

COLOR

Sets the color for new objects.

PROPERTIES

Controls properties of existing objects.

System Variables

CECOLOR

Sets the color of new objects.

Utilities

No entries

Command Modifiers

No entries

Work with Linetypes

You can use linetypes to distinguish objects from one another visually and make your drawing easier to read.

Overview of Linetypes

A linetype is a repeating pattern of dashes, dots, and blank spaces displayed in a line or a curve. You assign linetypes to objects either by layer or by specifying the linetype explicitly, independent of layers.

In addition to choosing a linetype, you can set its scale to control the size of the dashes and spaces, and you can create your own custom linetypes.

NOTE These linetypes should not be confused with the hardware linetypes provided by some plotters. The two types of dashed lines produce similar results. Do not use both types at the same time, however, because the results can be unpredictable.

See also:

- "Custom Linetypes" in the *Customization Guide*

Quick Reference

Commands

LINETYPE

Loads, sets, and modifies linetypes.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Load Linetypes

At the start of a project, you load the linetypes that are required for the project so that they are available when you need them.

If you want to know what linetypes are already available, you can display a list of linetypes that are loaded in the drawing or stored in an LIN (linetype definition) file.

This program includes the linetype definition files *acad.lin* and *acadiso.lin*. Which linetype file is appropriate depends on whether you use imperial or metric measurements.

- For imperial units, use the *acad.lin* file.
- For metric measurements, use the *acadiso.lin* file.

Both linetype definition files contain several complex linetypes.

If you select a linetype whose name begins with ACAD_ISO, you can use the ISO pen-width option when you plot.

You can remove unreferenced linetype information with PURGE or by deleting the linetype from the Linetype Manager. BYBLOCK, BYLAYER, and CONTINUOUS linetypes cannot be removed.

To load a linetype

1 Click Home tab ➤ Properties panel ➤ Linetype.

2 In the Linetype drop-down list, click Other. Then, in the Linetype Manager dialog box, click Load.

3 In the Load or Reload Linetypes dialog box, select a linetype. Click OK.

   If the linetype you need is not listed, click File. In the Select Linetype File dialog box, select an LIN file whose linetypes you want to list and click. The dialog box displays the linetype definitions stored in the selected LIN file. Select a linetype. Click OK.

   You can hold down CTRL to select several linetypes or SHIFT to select a range of linetypes.

4 Click OK.

Command entry: LINETYPE

To list the linetypes loaded in the current drawing

1 Click Home tab ➤ Properties panel ➤ Linetype.

2 Click anywhere outside the box to close it.

To list the linetypes in a linetype definition file

1 Click Home tab ➤ Properties panel ➤ Linetype.

2 In the Linetype drop-down list, click Other. Then, in the Linetype Manager dialog box, click Load.

3 In the Load or Reload Linetypes dialog box, click File.

4 In the Select Linetype File dialog box, select an LIN (linetype definition) file whose linetypes you want to lit. Click Open.

   The dialog box displays the linetype definitions stored in the selected LIN file.

5 In the Load or Reload Linetypes dialog box, click Cancel.

6 Click Cancel to close the Linetype Manager.

Command entry: LINETYPE

To unload an unused linetype

1 Click Home tab ➤ Properties panel ➤ Linetype. 

2 In the Linetype drop-down list, click Other. Then, in the Linetype Manager dialog box, select a linetype. Click Delete.

   The selected linetype is unloaded. Certain linetypes cannot be unloaded: BYLAYER, BYBLOCK, CONTINUOUS, and any linetypes currently in use.

Command entry: LINETYPE

To purge an unused linetype

1 Click Tools tab ➤ Drawing Utilities panel ➤ Purge. 

   The Purge dialog box displays a tree view of object types with items that can be purged.

2 To purge unreferenced linetypes, use one of the following methods:

   - To purge all unreferenced linetypes, select Linetypes.
   - To purge specific linetypes, double-click Linetypes to expand the tree view. Then select the linetypes to be purged.

   If the item you want to purge is not listed, select View Items You Cannot Purge.

3 You are prompted to confirm each item in the list. If you do not want to confirm each purge, clear the Confirm Each Item to Be Purged option.

4 Click Purge.

   To confirm the purging of each item, respond to the prompt by choosing Yes or No, or Yes to All if more than one item is selected.

5 Click Close.

Command entry: PURGE

Quick Reference

Commands

LINETYPE

Loads, sets, and modifies linetypes.

PURGE

Removes unused items, such as block definitions and layers, from the drawing.

RENAME

Changes the names assigned to items such as layers and dimension styles.

System Variables

MEASUREINIT

Controls whether a drawing you start from scratch uses imperial or metric default settings.

MEASUREMENT

Controls whether the current drawing uses imperial or metric hatch pattern and linetype files.

Utilities

No entries

Command Modifiers

No entries

Set the Current Linetype

All objects are created using the current linetype, which is displayed in the Linetype control on the Properties toolbar.

You can also set the current linetype with the Linetype control.

If the current linetype is set to BYLAYER, objects are created with the linetype assigned to the current layer.

If the current linetype is set to BYBLOCK, objects are created using the CONTINUOUS linetype until the objects are grouped into a block. When the block is inserted into the drawing, those objects acquire the current linetype setting.

If you do not want the current linetype to be the linetype assigned to the current layer, you can specify a different linetype explicitly.

The program does not display the linetype of certain objects: text, points, viewports, hatches, and blocks.

To set the linetype for all new objects

1 Click Home tab ➤ Properties panel ➤ Linetype. 

2 In the Linetype drop-down list, click Other. Then, in the Linetype Manager dialog box, click Load.

   You can hold down CTRL to select several linetypes or SHIFT to select a range of linetypes.

3 In the Linetype Manager dialog box, do one of the following:
   - Select a linetype and select Current to draw all new objects with that linetype.
   - Select BYLAYER to draw new objects in the linetype assigned to the current layer.
   - Select BYBLOCK to draw new objects in the current linetype until they are grouped into a block. When the block is inserted into a drawing, the objects in the block acquire the current linetype setting.

4 Click OK.

NOTE The Linetype control displays the current linetype. If the linetype you want to use is already loaded, you can click the Linetype control and click the linetype to make it current.

Command entry: LINETYPE

To change the linetype assigned to a layer

1 Click Home tab ➤ Layers panel ➤ Layer Properties. 

2. In the Layer Properties Manager, select the linetype name you want to change.
3. In the Select Linetype dialog box, select the linetype you want. Click OK.
4. Click OK again.

Toolbar: Layers 
Command entry: LAYER

Quick Reference

Commands

LINETYPE

Loads, sets, and modifies linetypes.

System Variables

CELTYPE

Sets the linetype of new objects.

Utilities

No entries

Command Modifiers

No entries

Change the Linetype of an Object

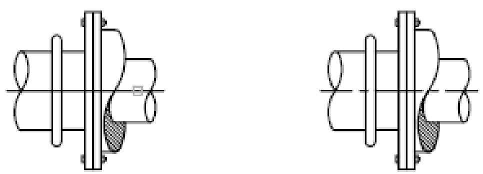
You can change the linetype of an object by reassigning it to another layer, by changing the linetype of the layer the object is on, or by specifying a linetype for the object explicitly.

You have three choices for changing the linetype of an object:

- Reassign the object to another layer with a different linetype. If an object's linetype is set to BYLAYER, and you reassign the object to a different layer, it acquires its linetype from the new layer.

- Change the linetype assigned to the layer that the object is on. If an object's linetype is set to BYLAYER, it acquires the linetype of its layer. When you change the linetype assigned to a layer, all objects on that layer assigned the BYLAYER linetype are updated automatically.
- Specify a linetype for an object to override the layer's linetype. You can specify the linetype of each object explicitly. If you want to override the layer-determined linetype of an object with a different one, change an existing object's linetype from BYLAYER to a specific linetype, such as DASHED.

If you want to set a specific linetype for all subsequently created objects, change the current linetype setting on the Properties toolbar from BYLAYER to a specific linetype.



object selected

result-continuous linetype changed to center

See also:

- Override Layer Properties in Viewports on page 559

To change the default linetype assigned to a layer

1. Click Home tab ➤ Layers panel ➤ Layer Properties. 
2. In the Layer Properties Manager, select the linetype name you want to change.
3. In the Select Linetype dialog box, select a linetype.

   If the linetype you need is not listed, click Load. Select a linetype and click OK, or click File to open additional linetype definition (LIN) files.
4. Click OK to exit each dialog box.

Toolbar: Layers 

Command entry: LAYER

To change the linetype of an object, overriding the layer's linetype

1 Select the objects whose linetype you want to change.

2 Click Home tab ➤ Palettes panel ➤ Properties. 

3 On the Properties palette, click the Linetype control.

4 Choose the linetype that you want to assign to the objects.

Command entry: PROPERTIES

Quick Reference

Commands

CHANGE

Changes the properties of existing objects.

CHPROP

Changes the properties of an object.

LAYER

Manages layers and layer properties.

LINETYPE

Loads, sets, and modifies linetypes.

PROPERTIES

Controls properties of existing objects.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Control Linetype Scale

You can use the same linetype at different scales by changing the linetype scale factor either globally or individually for each object.

By default, both global and individual linetype scales are set to 1.0. The smaller the scale, the more repetitions of the pattern are generated per drawing unit. For example, with a setting of 0.5, two repetitions of the pattern in the linetype definition are displayed for each drawing unit. Short line segments that cannot display one full linetype pattern are displayed as continuous. You can use a smaller linetype scale for lines that are too short to display even one dash sequence.

The Linetype Manager displays the Global Scale Factor and Current Object Scale.

- The Global Scale Factor value controls the LTSCALE system variable, which changes the linetype scale globally for both new and existing objects.
- The Current Object Scale value controls the CELTSCALE system variable, which sets the linetype scale for new objects.

The CELTSCALE value is multiplied by the LTSCALE value to get the displayed linetype scale. You can easily change linetype scales in your drawing either individually or globally.

In a layout, you can adjust the scaling of linetypes in different viewports with PSLTSCALE.

To change the linetype scale of selected objects

1. Select the objects whose linetype scale you want to change.

2 Click Home tab ➤ Palettes panel ➤ Properties. 
Alternatively, right-click one of the objects. Click Properties.

3 In the Properties palette, select Linetype Scale and enter the new value.

Toolbar: Standard 
Command entry: PROPERTIES

To set the linetype scale for new objects

1 Click Home tab ➤ Properties panel ➤ Linetype. 

2 In the Linetype drop-down list, select Other.

3 In the Linetype Manager, click Show Details to expand the dialog box.

4 Enter a new value for Current Object Scale.

5 Click OK.

Command entry: LINETYPE

To change linetype scale globally

1 Click Home tab ➤ Properties panel ➤ Linetype. 

2 In the Linetype Manager, click Show Details to expand the dialog box.

3 Enter a new value for Global Scale Factor.

4 Click OK.

Command entry: LINETYPE

Quick Reference

Commands

LINETYPE

Loads, sets, and modifies linetypes.

System Variables

CELTSCALE

Sets the current object linetype scaling factor.

LTSCALE

Sets the global linetype scale factor.

PSLTSCALE

Controls the linetype scaling of objects displayed in paper space viewports.

Utilities

No entries

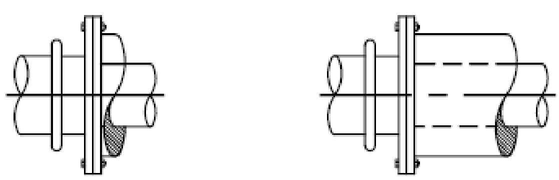
Command Modifiers

No entries

Display Linetypes on Short Segments and Polylines

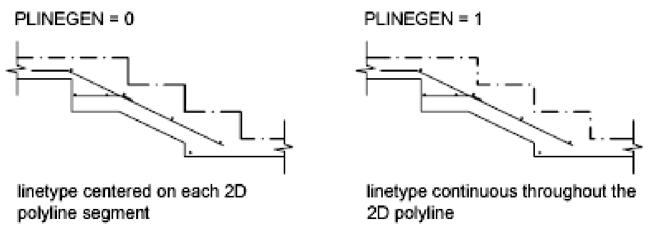
You can center the pattern of a linetype on each segment of a polyline, and you can control how the linetype is displayed on short segments.

If a line is too short to hold even one dash sequence, the result is a continuous line between the endpoints, as shown below.



You can accommodate short segments by using a smaller value for their individual linetype scales. For more information, see Control Linetype Scale on page 610.

For polylines, you can specify whether a linetype pattern is centered on each segment or is continuous across vertices throughout the entire length of the polyline. You do this by setting the PLINEGEN system variable.



To set the linetype display for all new polylines

1. At the Command prompt, enter **plinegen**.
2. Enter **1** to make the linetype pattern continue throughout the entire length of two-dimensional polylines, or enter **0** to center the linetype pattern on each segment.

Command entry: PLINEGEN

To change the linetype display of existing polylines

1. Select the polyline whose linetype display you want to change.
2. Click Home tab ➤ Palettes panel ➤ Properties. 
3. In the Properties palette, click Linetype Generation and select Enabled or Disabled.

Toolbar: Standard 
Command entry: PROPERTIES

Quick Reference

Commands

PROPERTIES

Controls properties of existing objects.

System Variables

PLINEGEN

Sets how linetype patterns generate around the vertices of a 2D polyline.

Utilities

No entries

Command Modifiers

No entries

Control Lineweights

You can control the thickness of an object's lines in both the drawing display and plotting.

Overview of Lineweights

Lineweights are width values that are assigned to graphical objects as well as some types of text.

Using lineweights, you can create heavy and thin lines to show cuts in sections, depth in elevations, dimension lines and tick marks, and differences in details. For example, by assigning varying lineweights to different layers, you can easily differentiate between new, existing, and demolition construction. Lineweights are not displayed unless the LWT button on the status bar is selected.

TrueType fonts, raster images, points, and solid fills (2D solids) cannot display lineweight. Wide polylines show lineweights only when displayed outside of the plan view. You can export drawings to other applications or cut objects to the Clipboard and retain lineweight information.

In model space, lineweights are displayed in pixels and do not change when zoomed in or out. Thus, you should not use lineweights to represent the exact width of an object in model space. For example, if you want to draw an object with a real-world width of 0.5 inches, do not use a lineweight; instead, use a polyline with a width of 0.5 inches to represent the object.

You can also plot objects in your drawing with custom lineweight values. Use the Plot Style Table Editor to adjust the fixed lineweight values to plot at a new value.

Lineweight Scale in Drawings

Objects with a lineweight are plotted with the exact width of the assigned lineweight value. The standard settings for these values include BYLAYER, BYBLOCK, and Default. They are displayed in either inches or millimeters, with millimeters being the default. All layers are initially set to 0.25 mm, controlled by the LWDEFAULT system variable.

A lineweight value of 0.025 mm or less is displayed as one pixel in model space and is plotted at the thinnest lineweight available on the specified plotting device. Lineweight values that you enter at the Command prompt are rounded to the nearest predefined value.

You set the lineweight units and the default value in the Lineweight Settings dialog box. You can access the Lineweight Settings dialog box by using the LWEIGHT command, by right-clicking the LWT button on the status bar and choosing Settings, or by choosing Lineweight Settings on the User Preferences tab in the Options dialog box.

See also:

- Draw Polylines on page 801

To assign width to polylines

1. Click Home tab ➤ Draw panel ➤ Polyline.
2. Specify the start point for the first segment in the polyline.
3. Enter **w** (width).
4. Enter a value for the width at the start of the line segment.
5. Enter a value for the width at the end of the line segment.
6. Specify the endpoint of the first segment of the polyline.
7. Enter **w** to specify different widths for the next segment, or press ENTER to end the command.

Toolbar: Draw
Command entry: PLINE

To assign a lineweight to a layer

1 Click Home tab ➤ Layers panel ➤ Layer Properties. 

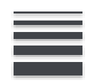
2 In the Layer Properties Manager, select a layer. Click the lineweight associated with that layer.

3 In the Lineweight dialog box, select a lineweight from the list.

4 Click OK to close each dialog box.

Command entry: LAYER

To set the display scale of lineweights on the Model tab

1 Click Home tab ➤ Properties panel ➤ Lineweight. 

2 In the Lineweight drop-down list, select Lineweight Settings.

3 In the Lineweight Settings dialog box, under Adjust Display Scale, move the slider to change the scale.

4 Click OK.

Command entry: LAYER, LWEIGHT

Quick Reference

Commands

LWEIGHT

Sets the current lineweight, lineweight display options, and lineweight units.

PLINE

Creates a 2D polyline.

System Variables

LWDEFAULT

Sets the value for the default lineweight.

LWDISPLAY

Controls whether the lineweights of objects are displayed.

LWUNITS

Controls whether lineweight units are displayed in inches or millimeters.

Utilities

No entries

Command Modifiers

No entries

Display Lineweights

Lineweights can be turned on and off in a drawing, and are displayed differently in model space than in a paper space layout.

- In model space, a 0-value lineweight is displayed as one pixel, and other lineweights use a pixel width proportional to their real-unit value.
- In a paper space layout, lineweights are displayed in the exact plotting width.

Regeneration time increases with lineweights that are represented by more than one pixel. Turn off the display of lineweights to optimize performance of the program.

You can turn the display of lineweights on or off by clicking LWT on the status bar. This setting does not affect the plotting of lineweights.

Display Lineweights in Model Space

Lineweight display in model space does not change with the zoom factor. For example, a lineweight value that is represented by a width of four pixels is always displayed using four pixels regardless of how far you zoom in. If you want the lineweights on objects to appear thicker or thinner on the Model tab, use LWEIGHT to set their display scale. Changing the display scale does not affect the lineweight plotting value.

In model space, weighted lines that are joined form a beveled joint with no end caps. You can use plot styles to apply different joins and endcap styles to objects with lineweights.

NOTE Different styles of endcaps and joins of objects with lineweight are displayed only in a full plot preview.

Display Lineweights in Layouts

In layouts and plot preview, lineweights are displayed in real-world units, and lineweight display changes with the zoom factor. You can control lineweight plotting and scaling in your drawing in the Plot dialog box, Plot Settings tab.

To display or hide lineweights

Use one of the following methods:

- Click LWT on the status bar.
- Select or clear Display Lineweight in the Lineweight Settings dialog box.
- Set the LWDISPLAY system variable to 0 or 1.

Command entry: LWEIGHT

Quick Reference

Commands

LAYER

Manages layers and layer properties.

LWEIGHT

Sets the current lineweight, lineweight display options, and lineweight units.

PEDIT

Edits polylines and 3D polygon meshes.

PLINE

Creates a 2D polyline.

PLOT

Plots a drawing to a plotter, printer, or file.

System Variables

LWDEFAULT

Sets the value for the default lineweight.

LWDISPLAY

Controls whether the lineweights of objects are displayed.

LWUNITS

Controls whether lineweight units are displayed in inches or millimeters.

PLINEWID

Stores the default polyline width.

Utilities

No entries

Command Modifiers

No entries

Set the Current Lineweight

The current lineweight is the lineweight used for any objects you draw until you make another lineweight current.

All objects are created using the current lineweight, which is displayed in the Lineweight control on the Properties toolbar. You can also set the current lineweight with the Lineweight control.

If the current lineweight is set to BYLAYER, objects are created with the lineweight assigned to the current layer.

If the current lineweight is set to BYBLOCK, objects are created using the default lineweight setting until the objects are grouped into a block. When the block is inserted into the drawing, it acquires the current lineweight setting.

If you do not want the current lineweight to be the lineweight assigned to the current layer, you can specify a different lineweight explicitly.

Objects in drawings created in an earlier release of AutoCAD are assigned the lineweight value of BYLAYER, and all layers are set to DEFAULT. Lineweight assigned to objects is displayed as a solid fill drawn in the object's assigned color.

To make a lineweight current for creating objects

1 Click Home tab ➤ Properties panel ➤ Lineweight.

2 In the Lineweight drop-down list, select Lineweight Settings.

3 In the Lineweight Settings dialog box, select a lineweight.

4 Click OK.

NOTE To display the lineweight at its current setting, the Display Lineweight option must be selected in the Display Lineweight dialog box.

Command entry: LAYER, LWEIGHT

Quick Reference

Commands

LAYER

Manages layers and layer properties.

LWEIGHT

Sets the current lineweight, lineweight display options, and lineweight units.

PEDIT

Edits polylines and 3D polygon meshes.

PLINE

Creates a 2D polyline.

PLOT

Plots a drawing to a plotter, printer, or file.

System Variables

LWDEFAULT

Sets the value for the default lineweight.

LWDISPLAY

Controls whether the lineweights of objects are displayed.

LWUNITS

Controls whether lineweight units are displayed in inches or millimeters.

PLINEWID

Stores the default polyline width.

Utilities

No entries

Command Modifiers

No entries

Change the Lineweight of an Object

You can change the lineweight of an object by reassigning it to another layer, by changing the lineweight of the layer the object is on, or by specifying a lineweight for the object explicitly.

You have three choices for changing the lineweight of an object:

- Reassign the object to another layer with a different lineweight. If an object's lineweight is set to BYLAYER, and you reassign the object to a different layer, it acquires its lineweight from the new layer.
- Change the lineweight assigned to the layer that the object is on. If an object's lineweight is set to BYLAYER, it acquires the lineweight of its layer. When you change the lineweight assigned to a layer, all objects on that layer assigned the BYLAYER lineweight are updated automatically.
- Specify a lineweight for an object to override the layer's lineweight. You can specify the lineweight of each object explicitly. If you want to override the layer-determined lineweight of an object with a different one, change an existing object's lineweight from BYLAYER to a specific lineweight.

If you want to set a specific lineweight for all subsequently created objects, change the current lineweight setting on the Properties toolbar from BYLAYER to a specific lineweight.

See also:

- Override Layer Properties in Viewports on page 559

To change the line width of a polyline, donut, rectangle, or polygon

1 Click Home tab ➤ Modify panel ➤ Edit Polyline.
2 Select one or more polyline objects.
3 Enter **w** (Width) and enter a new width for all segments.
4 Press ENTER to end the command.

Toolbar: Modify II
Command entry: PEDIT

Quick Reference

Commands

LAYER

Manages layers and layer properties.

LWEIGHT

Sets the current lineweight, lineweight display options, and lineweight units.

PEDIT

Edits polylines and 3D polygon meshes.

PLINE

Creates a 2D polyline.

PLOT

Plots a drawing to a plotter, printer, or file.

System Variables

LWDEFAULT

Sets the value for the default lineweight.

LWDISPLAY

Controls whether the lineweights of objects are displayed.

LWUNITS

Controls whether lineweight units are displayed in inches or millimeters.

PLINEWID

Stores the default polyline width.

Utilities

No entries

Command Modifiers

No entries

Control the Display Properties of Certain Objects

You can control how overlapping objects and certain other objects are displayed and plotted.

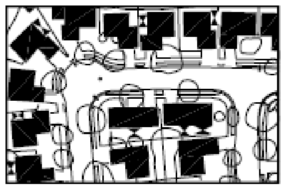
Control the Display of Polylines, Hatches, Gradient Fills, Lineweights, and Text

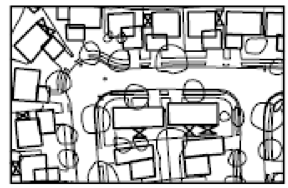
You can simplify the display of certain kinds of objects in order to speed performance.

Display performance is improved when wide polylines and donuts, solid-filled polygons (two-dimensional solids), hatches, gradient fills, and text are displayed in simplified form. Simplified display also increases the speed of creating test plots.

Turn Off Solid Fill

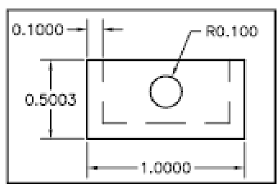
When you turn off Fill mode, wide polylines, solid-filled polygons, gradient fill, and hatches are displayed in outline form. Except for patterned hatches and gradient fills, solid fill is automatically turned off for hidden view and nonplan views in three dimensions.


Fill mode on

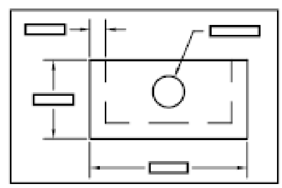

Fill mode off

Use Quick Text

When you turn on Quick Text mode in drawings that contain a lot of text using complex fonts, only a rectangular frame defining the text is displayed or plotted.



Quick Text mode off


Quick Text mode on

Turn Off Lineweights

Any lineweight width that is represented by more than one pixel may slow down performance. If you want to improve display performance, turn lineweights off. You can turn lineweights on and off by choosing the LWT button on the status bar or by using the Lineweight Settings dialog box. Lineweights are always plotted at their real-world value whether their display is turned on or off.

Update the Display

New objects automatically use the current settings for displays of solid fill and text. Except for lineweights, to update the display of existing objects using these settings, you must use REGEN.

See also:

- Use Layers to Manage Complexity on page 535
- Display Lineweights on page 617
- Use TrueType Fonts on page 1549

To turn the display of solid fill on or off

1 Click Tools menu ➤ Options. 

2 In the Options dialog box, Display tab, under Display Performance, select Apply Solid Fill.

   A check mark indicates that Fill mode is on.

3 Click OK.

4 To display your changes, click View menu ➤ Regen.

Command entry: OPTIONS, REGEN

To turn the display of text on or off

1 Click Tools ➤ Options. 

2 In the Options dialog box, Display tab, under Display Performance, select Show Text Boundary Frame Only.

   The check mark indicates that text is displayed as a rectangular frame.

3 Click OK.

4 To display your changes, click View menu ➤ Regen.

Command entry: OPTIONS, REGEN

To turn lineweights on or off

1 Click Home tab ➤ Properties panel ➤ Lineweight. 

2 In the Lineweight drop-dow list, select Lineweight Settings.

3 In the Lineweight Settings dialog box, select or clear Display Lineweight.

4 Click OK.

Command entry: LWEIGHT

Quick Reference

Commands

DSETTINGS

Sets grid and snap, polar and object snap tracking, object snap modes, Dynamic Input, and Quick Properties.

FILL

Controls the filling of objects such as hatches, 2D solids, and wide polylines.

LWEIGHT

Sets the current lineweight, lineweight display options, and lineweight units.

QTEXT

Controls the display and plotting of text and attribute objects.

REGEN

Regenerates the entire drawing from the current viewport.

System Variables

FILLMODE

Specifies whether hatches and fills, 2D solids, and wide polylines are filled in.

LWDISPLAY

Controls whether the lineweights of objects are displayed.

QTEXTMODE

Controls how text is displayed.

TEXTFILL

Controls the filling of TrueType fonts while plotting and rendering.

TEXTQLTY

Sets the resolution tessellation fineness of text outlines.

Utilities

No entries

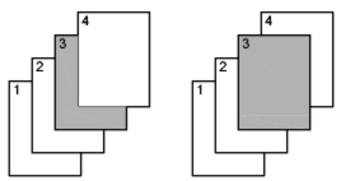
Command Modifiers

No entries

Control How Overlapping Objects Are Displayed

You can control which overlapping objects appear to be on top.

Generally, overlapping objects such as text, wide polylines, and solid-filled polygons are displayed in the order they are created: newly created objects in front of existing objects. You can use DRAWORDER to change the draw order (which is the display and plotting order) of any objects. TEXTTOFRONT changes the draw order of all text and dimensions in the drawing.



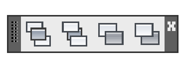
The rectangles are displayed in the order they were created

The third rectangle has an assigned draw order

NOTE Overlapping objects cannot be controlled between model space and paper space. They can be controlled only within the same space.

To change the draw order of overlapping objects

1. Click Home tab ➤ Modify panel ➤ Draw Order drop-down list.
2. From the draw order drop down list, click one of the options.
3. Select the object(s) whose draw order you want to modify and press ENTER.
4. Select the reference object(s) and press ENTER. (This step is necessary only for the Bring Above Objects and Send Under Objects options.)

Toolbar: Draw Order 
Command entry: DRAWORDER
Shortcut menu: Select an object and right-click. Click Draw Order

Quick Reference

Commands

DRAWORDER

Changes the draw order of images and other objects.

QSELECT

Creates a selection set based on filtering criteria.

REGEN

Regenerates the entire drawing from the current viewport.

SELECT

Places selected objects in the Previous selection set.

TEXTTOFRONT

Brings text and dimensions in front of all other objects in the drawing.

WBLOCK

Writes objects or a block to a new drawing file.

System Variables

DRAWORDERCTL

Controls the default display behavior of overlapping objects when they are created or edited.

HPDRAWORDER

Controls the draw order of hatches and fills.

SORTENTS

Controls object sorting in support of draw order for several operations.

Utilities

No entries

Command Modifiers

No entries

Use Precision Tools

16

You can use a variety of precision drawing tools to help you produce accurate drawings quickly and without performing tedious calculations.

Use Coordinates and Coordinate Systems (UCS)

For precise coordinate input, you can use several coordinate system entry methods. You can also use a movable coordinate system, the user coordinate system (UCS), for convenient coordinate entry and to establish workplanes.

Overview of Coordinate Entry

When a command prompts you for a point, you can use the pointing device to specify a point, or you can enter a coordinate value at the command prompt. When dynamic input is on, you can enter coordinate values in tooltips near the cursor. You can enter two-dimensional coordinates as either Cartesian (*X,Y*) or polar coordinates.

Cartesian and Polar Coordinates

A Cartesian coordinate system has three axes, *X*, *Y*, and *Z*. When you enter coordinate values, you indicate a point's distance (in units) and its direction (+ or -) along the *X*, *Y*, and *Z* axes relative to the coordinate system origin (0,0,0).

In 2D, you specify points on the *XY* plane, also called the *workplane*. The workplane is similar to a flat sheet of grid paper. The *X* value of a Cartesian coordinate specifies horizontal distance, and the *Y* value specifies vertical distance. The origin point (0,0) indicates where the two axes intersect.

Polar coordinates use a distance and an angle to locate a point. With both Cartesian and polar coordinates, you can enter absolute coordinates based on the origin (0,0), or relative coordinates based on the last point specified.

Another method of entering a relative coordinate is by moving the cursor to specify a direction and then entering a distance directly. This method is called direct distance entry.

You can enter coordinates in scientific, decimal, engineering, architectural, or fractional notation. You can enter angles in grads, radians, surveyor's units, or degrees, minutes, and seconds. The UNITS command controls unit format.

Display Coordinates on the Status Bar

The current cursor location is displayed as a coordinate value on the status bar.

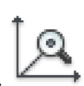
There are three types of coordinate display: static, dynamic, and distance and angle.

- **Static display**. Updates only when you specify a point.
- **Dynamic display.** Updates as you move the cursor.
- **Distance and angle display.** Updates the relative distance *(distance<angle)* as you move the cursor. This option is available only when you draw lines or other objects that prompt for more than one point.

See also:

- Enter Cartesian Coordinates on page 632
- Enter Polar Coordinates on page 635
- Enter 3D Coordinates on page 637
- Use Dynamic Input on page 668

To display the coordinate values of a point

1. Click Tools tab ➤ Inquiry panel ➤ ID Point. 
2. Select the location you want to identify.
   The *X,Y,Z* coordinate values are displayed at the command prompt.

Toolbar: Inquiry
Command entry: ID

To visually locate a point

1. Click Tools tab ➤ Inquiry panel ➤ ID Point.
2. At the command prompt, enter the coordinate values of the point you want to locate.

   If the BLIPMODE system variable is on, a blip (a small cross) is displayed at the point location.

Toolbar: Inquiry
Command entry: ID

To change the coordinate display on the status bar

Use one of the following methods:

- Click the coordinate display at the Specify Next Point prompt.
- Press CTRL+L.
- Set the COORDS system variable to 0 for static display, 1 for dynamic display, or 2 for distance and angle display.

Quick Reference

Commands

BLIPMODE

Controls the display of marker blips.

ID

Displays the UCS coordinate values of a specified location.

LIST

Displays property data for selected objects.

System Variables

BLIPMODE

Controls the display of marker blips.

COORDS

Controls the format and update frequency of coordinates on the status line.

LASTPOINT

Stores the last point specified, expressed as UCS coordinates for the current space.

Utilities

No entries

Command Modifiers

No entries

Enter 2D Coordinates

Absolute and relative 2D Cartesian and polar coordinates determine precise locations of objects in a drawing.

Enter Cartesian Coordinates

You can use absolute or relative Cartesian (rectangular) coordinates to locate points when creating objects.

To use Cartesian coordinates to specify a point, enter an *X* value and a *Y* value separated by a comma *(X,Y)*. The *X* value is the positive or negative distance, in units, along the horizontal axis. The *Y* value is the positive or negative distance, in units, along the vertical axis.

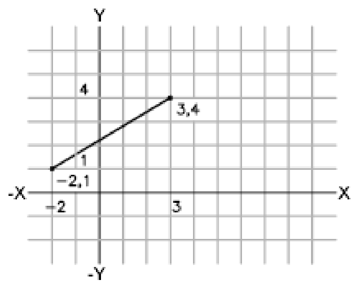
Absolute coordinates are based on the UCS origin (0,0), which is the intersection of the *X* and *Y* axes. Use absolute coordinates when you know the precise *X* and *Y* values of the point.

With dynamic input, you can specify absolute coordinates with the # prefix. If you enter coordinates on the command line instead of in the tooltip, the # prefix is not used. For example, entering **#3,4** specifies a point 3 units along the *X* axis and 4 units along the *Y* axis from the UCS origin. For more information about dynamic input, see Use Dynamic Input on page 668.

The following example draws a line beginning at an *X* value of -2, a *Y* value of 1, and an endpoint at 3,4. Enter the following in the tooltip:

Command: **line**
From point: #**-2,1**
To point: #**3,4**

The line is located as follows:

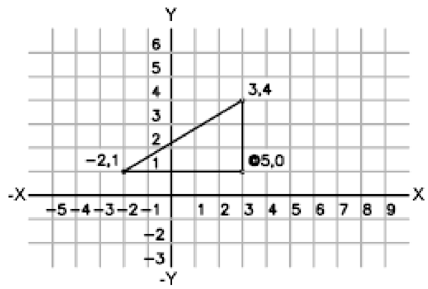


Relative coordinates are based on the last point entered. Use relative coordinates when you know the location of a point in relation to the previous point.

To specify relative coordinates, precede the coordinate values with an @ sign. For example, entering **@3,4** specifies a point 3 units along the *X* axis and 4 units along the *Y* axis from the last point specified.

The following example draws the sides of a triangle. The first side is a line starting at the absolute coordinates -2,1 and ending at a point 5 units in the *X* direction and 0 units in the *Y* direction. The second side is a line starting at the endpoint of the first line and ending at a point 0 units in the *X* direction and 3 units in the *Y* direction. The final line segment uses relative coordinates to return to the starting point.

Command: **line**
From point: #**-2,1**
To point: **@5,0**
To point: **@0,3**
To point: **@-5,-3**



To enter absolute Cartesian coordinates (2D)

- At a prompt for a point, enter coordinates in the tooltip using the following format:
  #*x*,*y*

  If dynamic input is turned off, enter coordinates on the command line using the following format:

  x,*y*

To enter relative Cartesian coordinates (2D)

- At a prompt for a point, enter coordinates using the following format:
  @*x*,*y*

Quick Reference

Commands

No entries

System Variables

COORDS

Controls the format and update frequency of coordinates on the status line.

Utilities

No entries

Command Modifiers

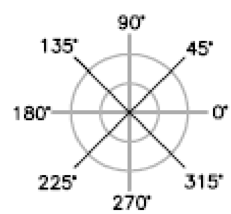
No entries

Enter Polar Coordinates

You can use absolute or relative polar coordinates (distance and angle) to locate points when creating objects.

To use polar coordinates to specify a point, enter a distance and an angle separated by an angle bracket (<).

By default, angles increase in the counterclockwise direction and decrease in the clockwise direction. To specify a clockwise direction, enter a negative value for the angle. For example, entering **1<315** locates the same point as entering **1<-45**. You can change the angle conventions for the current drawing with UNITS.

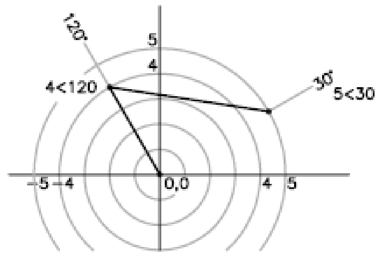


Absolute polar coordinates are measured from the UCS origin (0,0), which is the intersection of the *X* and *Y* axes. Use absolute polar coordinates when you know the precise distance and angle coordinates of the point.

With dynamic input, you can specify absolute coordinates with the # prefix. If you enter coordinates on the command line instead of in the tooltip, the # prefix is not used. For example, entering **#3<45** specifies a point 3 units from the origin at an angle of 45 degrees from the *X* axis. For more information about dynamic input, see Use Dynamic Input on page 668.

The following example shows two lines drawn with absolute polar coordinates using the default angle direction setting. Enter the following in the tooltip:

Command: **line**
From point: **#0,0**
To point: **#4<120**
To point: **#5<30**

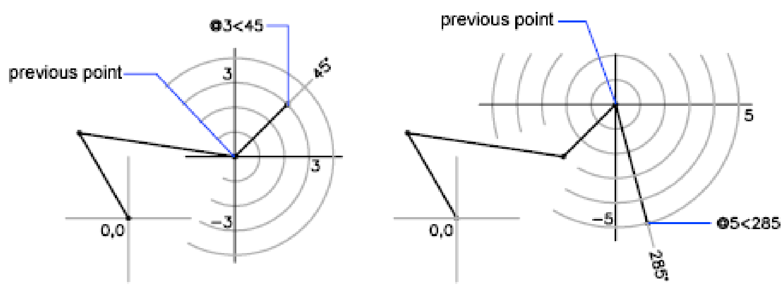


Relative coordinates are based on the last point entered. Use relative coordinates when you know the location of a point in relation to the previous point.

To specify relative coordinates, precede the coordinate values with an @ sign. For example, entering **@1<45** specifies a point at a distance of 1 unit from the last point specified at an angle of 45 degrees from the *X* axis.

The following example shows two lines drawn with relative polar coordinates. In each illustration, the line begins at the location labeled as the previous point.

Command: **line**
From point: **@3<45**
To point: **@5<285**



To enter absolute polar coordinates (2D)

- At a prompt for a point, enter coordinates in the tooltip using the following format:
  #distance<angle

  If dynamic input is turned off, enter coordinates on the command line using the following format:

  distance<angle

To enter relative polar coordinates (2D)

- At a prompt for a point, enter coordinates using the following format: ***@****distance<angle*

Quick Reference

Commands

UNITS

Controls coordinate and angle display formats and precision.

System Variables

COORDS

Controls the format and update frequency of coordinates on the status line.

Utilities

No entries

Command Modifiers

No entries

Enter 3D Coordinates

Cartesian, cylindrical, or spherical coordinates locate points when you are creating objects in 3D.

Enter 3D Cartesian Coordinates

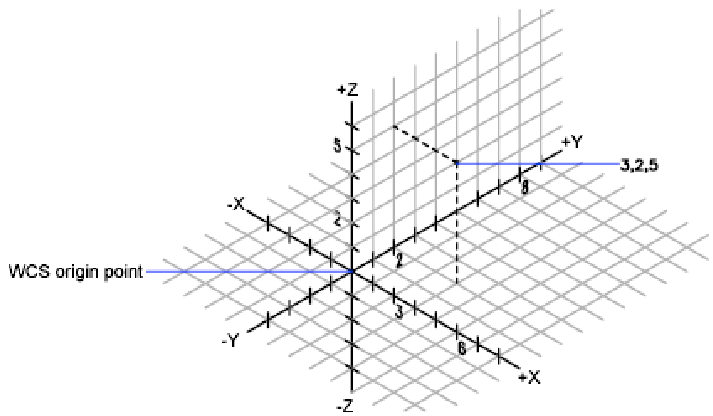
3D Cartesian coordinates specify a precise location by using three coordinate values: *X*, *Y*, and *Z*.

Entering 3D Cartesian coordinate values *(X,Y,Z)* is similar to entering 2D coordinate values *(X,Y)*. In addition to specifying *X* and *Y* values, you also specify a *Z* value using the following format:

X,Y,Z

NOTE For the following examples, it is assumed that dynamic input is turned off or that the coordinates are entered on the command line. With dynamic input, you specify absolute coordinates with the # prefix.

In the illustration below, the coordinate values of 3,2,5 indicates a point 3 units along the positive *X* axis, 2 units along the positive *Y* axis, and 5 units along the positive *Z* axis.



Use Default Z Values

When you enter coordinates in the format *X,Y*, the *Z* value is copied from the last point you entered. As a result, you can enter one location in the *X,Y,Z* format and then enter subsequent locations using the *X,Y* format with the *Z* value remaining constant. For example, if you enter the following coordinates for a line

From point: **0,0,5**
To point: **3,4**

both endpoints of the line will have a *Z* value of 5. When you begin or open any drawing, the initial default value of *Z* is greater than 0.

Use Absolute and Relative Coordinates

As with 2D coordinates, you can enter absolute coordinate values, which are based on the origin, or you can enter relative coordinate values, which are based on the last point entered. To enter relative coordinates, use the @ sign as a prefix. For example, use @**1,0,0** to enter a point one unit in the positive *X* direction from the previous point. To enter absolute coordinates at the command prompt, no prefix is necessary.

Digitize Coordinates

When you enter coordinates by digitizing, the UCS *Z* value for all coordinates is 0. You can use ELEV to set a default height above or below the *Z* = 0 plane for digitizing without moving the UCS.

To enter absolute coordinates (3D)

- At a prompt for a point, enter coordinates in the tooltip using the following format:
  #*x*,*y*,*z*

  If dynamic input is turned off, enter coordinates on the command line using the following format:

  x,*y*,*z*

To enter relative coordinates (3D)

- At a prompt for a point, enter coordinates using the following format:
  @*x*,*y*,*z*

Quick Reference

Commands

GRID

Displays a grid pattern in the current viewport.

SNAP

Restricts cursor movement to specified intervals.

TABLET

Calibrates, configures, and turns on and off an attached digitizing tablet.

UCS

Manages user coordinate systems.

UCSICON

Controls the visibility and placement of the UCS icon.

UNITS

Controls coordinate and angle display formats and precision.

System Variables

COORDS

Controls the format and update frequency of coordinates on the status line.

ELEVATION

Stores the current elevation of new objects relative to the current UCS.

TABMODE

Controls the use of the tablet.

Utilities

No entries

Command Modifiers

No entries

Enter Cylindrical Coordinates

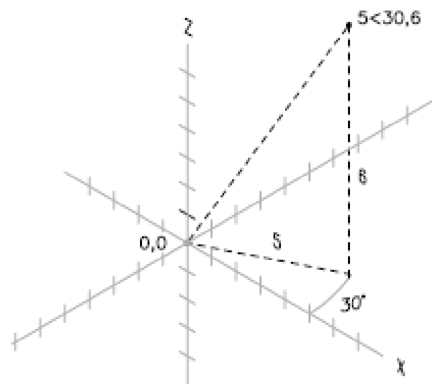
3D cylindrical coordinates describe a precise location by a distance from the UCS origin in the *XY* plane, an angle from the *X* axis in the *XY* plane, and a *Z* value.

Cylindrical coordinate entry is the 3D equivalent of 2D polar coordinate entry. It specifies an additional coordinate on an axis that is perpendicular to the *XY* plane. Cylindrical coordinates define points by a distance in the *XY* plane from the UCS origin, an angle from the *X* axis in the *XY* plane, and a *Z* value. You specify a point using *absolute* cylindrical coordinates with the following syntax:

X<[**angle from X axis**],**Z**

NOTE For the following examples, it is assumed that dynamic input is turned off or that the coordinates are entered on the command line. With dynamic input, you specify absolute coordinates with the # prefix.

In the illustration below, 5<30,6 indicates a point 5 units from the origin of the current UCS, 30 degrees from the *X* axis in the *XY* plane, and 6 units along the *Z* axis.



When you need to define a point based on a previous point rather than the UCS origin, you can enter *relative* cylindrical coordinate values with the @ prefix. For example, @4<45,5 specifies a point 4 units in the *XY* plane from the last point entered, at an angle of 45 degrees from the positive *X* direction, and extending 5 units in the positive *Z* direction.

To enter relative cylindrical coordinates

- At a prompt for a point, enter the coordinate values using the following format:
  @*x***<***angle from the X axis***,***z*

For example, @4<60,2 represents a location that is 4 units along the *X* axis from the last point measured at 60 degrees from the positive *X* axis and at 2 units in the positive *Z* direction.

Quick Reference

Commands

UNITS

Controls coordinate and angle display formats and precision.

System Variables

COORDS

Controls the format and update frequency of coordinates on the status line.

Utilities

No entries

Command Modifiers

No entries

Enter Spherical Coordinates

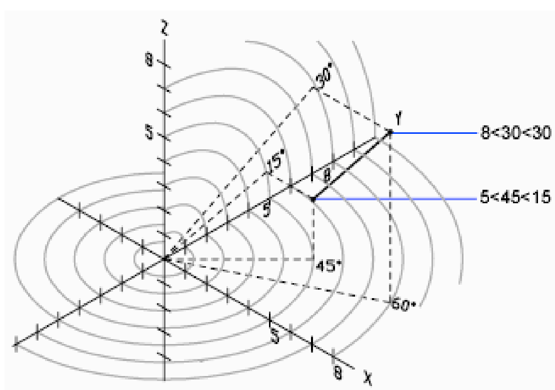
3D spherical coordinates specify a location by a distance from the origin of the current UCS, an angle from the *X* axis in the *XY* plane, and an angle from the *XY* plane.

Spherical coordinate entry in 3D is similar to polar coordinate entry in 2D. You locate a point by specifying its distance from the origin of the current UCS, its angle from the *X* axis (in the *XY* plane), and its angle from the *XY* plane, each angle preceded by an open angle bracket (<) as in the following format:

X<[**angle from X axis**]<[**angle from XY plane**]

NOTE For the following examples, it is assumed that dynamic input is turned off or that the coordinates are entered on the command line. With dynamic input, you specify absolute coordinates with the # prefix.

In the following illustration, 8<60<30 indicates a point 8 units from the origin of the current UCS in the *XY* plane, 60 degrees from the *X* axis in the *XY* plane, and 30 degrees up the *Z* axis from the *XY* plane. 5<45<15 indicates a point 5 units from the origin, 45 degrees from the *X* axis in the *XY* plane, and 15 degrees up from the *XY* plane.



When you need to define a point based on a previous point, enter the relative spherical coordinate values by preceding them with the @ sign.

To enter relative spherical coordinates

- At a prompt for a point, enter the coordinate values using the following format:
  @x<angle from the x axis<angle from the xy plane

For example, @4<60<30 represents a location that is 4 units from the last point measured at 60 degrees from the positive *X* axis in the *XY* plane and at 30 degrees from the *XY* plane.

Quick Reference

Commands

UNITS

Controls coordinate and angle display formats and precision.

System Variables

COORDS

Controls the format and update frequency of coordinates on the status line.

Utilities

No entries

Command Modifiers

No entries

Understand the User Coordinate System (UCS)

You can relocate and rotate the user coordinate system for convenient coordinate entry, grid display, grid snap, Ortho mode, and other drawing tools.

Understand the World and User Coordinate Systems

There are two coordinate systems: a fixed system called the world coordinate system (WCS) and a movable system called the user coordinate system (UCS). By default, these two systems are coincident in a new drawing.

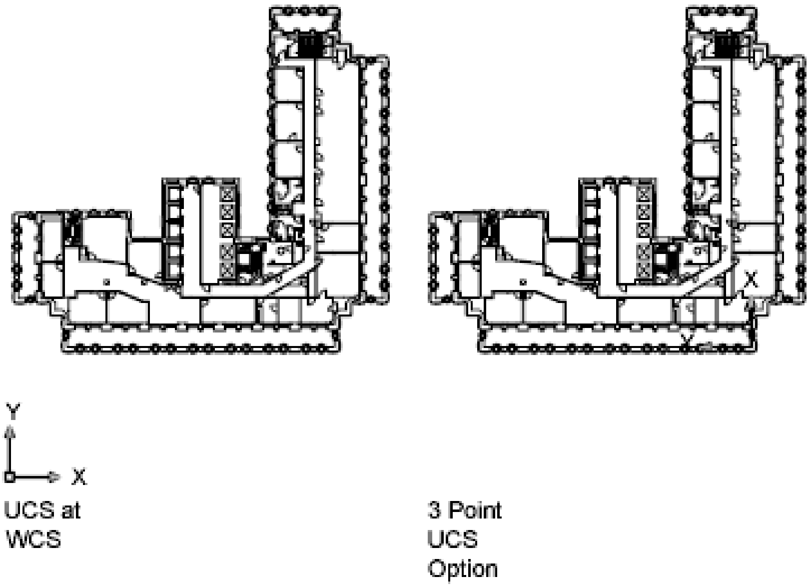
Normally in 2D views, the WCS *X* axis is horizontal and the *Y* axis is vertical. The WCS *origin* is where the *X* and *Y* axes intersect (0,0). All objects in a drawing file are defined by their WCS coordinates. However, it is usually more convenient to create and edit objects based on the movable UCS.

Work with the User Coordinate System

Virtually all coordinate entry as well as many other tools and operations reference the current UCS. 2D tools and operations that depend on the location and orientation of the UCS include the following:

- Absolute and relative coordinate entry
- Absolute reference angles
- Definition of horizontal and vertical for Ortho mode, polar tracking, object snap tracking, grid display, and grid snap
- Orientation of horizontal and vertical dimensions
- Orientation of text objects
- View rotation using the PLAN command

Moving or rotating the UCS can make it easier to work on particular areas of a drawing.



You can relocate the user coordinate system with methods such as the following:

- Move the UCS by defining a new origin point.
- Align the UCS with an existing object.
- Rotate the UCS by specifying a new origin point and a point on the new *X* axis.
- Rotate the current UCS a specified angle around the *Z* axis.
- Revert to the previous UCS.
- Restore the UCS to be coincident with the WCS.

Each of these methods have a corresponding option in the UCS command. Once you have defined a UCS, you can name it and then restore it when you need to use it again.

To define a new UCS origin in 2D

1 Click View tab ➤ UCS panel ➤ Origin.

2 Specify a point for the new origin.
The UCS origin (0,0) is redefined at the point you specify.

Toolbar: UCS
Command entry: UCS

To change the rotation angle of the UCS

1 Click View tab ➤ UCS panel ➤ Z.

2 Specify a rotation angle.

Toolbar: UCS
Command entry: UCS

To restore the UCS to be coincident with the WCS

1 Click View tab ➤ UCS panel ➤ Named UCS.

2 In the UCS dialog box, Named UCSs tab, select World.

3 Click Set Current.

4 Click OK.

Toolbar: UCS II
Command entry: UCSMAN

To restore the previous UCS

1 Click View tab ➤ UCS panel ➤ Named UCS.

2 In the UCS dialog box, Named UCSs tab, select Previous.

3 Click Set Current.

4 Click OK.

Toolbar: UCS II 
Command entry: UCSMAN

To save a UCS

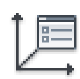
1 Click View tab ➤ UCS panel ➤ Named UCS.

   The new UCS is displayed in the UCS list as UNNAMED.

2 In the UCS dialog box, Named UCSs tab, select UNNAMED and enter a new name. (You can also select UNNAMED, and right-click. Click Rename.)

3 Click OK.

   You can use up to 255 characters, including letters, digits, and the special characters dollar sign ($), hyphen (-), and underscore (_). All UCS names are converted to uppercase.

Toolbar: UCS II 
Command entry: UCSMAN

To restore a named UCS

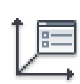
1 Click View tab ➤ UCS panel ➤ Named UCS.

2 In the UCS dialog box, Named UCSs tab, you can view the origin and axis direction of a listed UCS. Select the UCS name. Click Details.

   When you have finished viewing the list, click OK to return to the UCS dialog box.

3 Select the coordinate system you want to restore. Click Set Current.

4 Click OK.

Toolbar: UCS II 
Command entry: UCSMAN

To rename a UCS

1 Click View tab ➤ UCS panel ➤ Named UCS.

2 In the UCS dialog box, Named UCSs tab, select the coordinate system you want to rename. (You can also select UNNAMED, and right-click. Click Rename.)

3 Enter a new name.

4 Click OK.

Toolbar: UCS II 
Command entry: UCSMAN

To delete a named UCS

1 Click View tab ➤ UCS panel ➤ Named UCS.

2 In the UCS dialog box, Named UCSs tab, select the UCS you want to delete.

3 Press DELETE.

You cannot delete the current UCS or a UCS with the default name UNNAMED.

Toolbar: UCS II 
Command entry: UCSMAN

Quick Reference

Commands

UCS

Manages user coordinate systems.

UCSICON

Controls the visibility and placement of the UCS icon.

UCSMAN

Manages defined user coordinate systems.

System Variables

ERRNO

Displays the number of the appropriate error code when an AutoLISP function call causes an error that AutoCAD detects.

PUCSBASE

Stores the name of the UCS that defines the origin and orientation of orthographic UCS settings in paper space only.

UCSFOLLOW

Generates a plan view whenever you change from one UCS to another.

UCSNAME

Stores the name of the current coordinate system for the current viewport in the current space.

UCSORG

Stores the origin point of the current coordinate system for the current viewport in the current space.

UCSORTHO

Determines whether the related orthographic UCS setting is restored automatically when an orthographic view is restored.

UCSXDIR

Stores the X direction of the current UCS for the current viewport in the current space.

UCSYDIR

Stores the Y direction of the current UCS for the current viewport in the current space.

Utilities

No entries

Command Modifiers

No entries

Specify Workplanes in 3D (UCS)

Control of the user coordinate system is essential for effective 3D modeling.

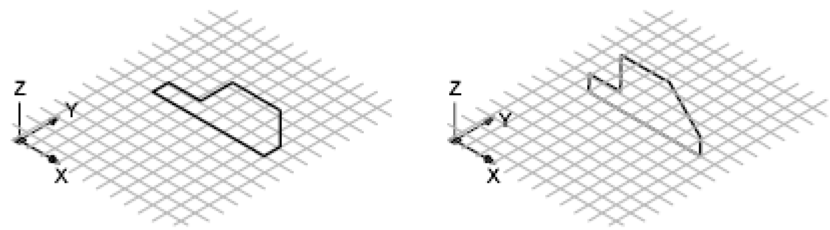
Understand the User Coordinate System in 3D

When you work in 3D, the user coordinate system is useful for entering coordinates, creating 3D objects on 2D workplanes, and rotating objects in 3D.

When you create or modify objects in a 3D environment, you can move and reorient the UCS in 3D model space to simplify your work. The *XY* plane of the UCS is called the *workplane*.

Important operations on objects in a 3D environment that depend on the location and orientation of the UCS include the following:

- Establish the workplane in on which to create and modify objects
- Establish the workplane that contains the grid display and grid snap
- Establish a new UCS *Z* axis about which to rotate objects in 3D
- Determine up and down directions as well as horizontal and vertical for Ortho mode, polar tracking, and object snap tracking
- Define a 3D view directly into the workplane with the PLAN command



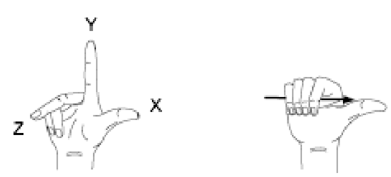
2D object on a 3D plane

2D object rotated on x-axis

Apply the Right-Hand Rule

Use the right-hand rule to determine the positive axis direction of the *Z* axis when you know the direction of the *X* and *Y* axes in a 3D coordinate system. Place the back of your right hand near the screen and point your thumb in the direction of the positive *X* axis. Extend your index and middle fingers as illustrated, pointing your index finger in the direction of the positive *Y* axis. Your middle finger indicates the direction of the positive *Z* axis. By rotating your hand, you see how the *X*, *Y*, and *Z* axes rotate as you change the UCS.

You can also use the right-hand rule for determining the default positive direction of rotation about an axis in 3D space. Point your right thumb in the positive direction of the axis and curl your fingers. Your fingers indicate the positive rotation direction about the axis.



NOTE By default, when you specify a view in 3D, it is established relative to the fixed WCS rather than the movable UCS.

To enter coordinates relative to the WCS

- Precede coordinate values with an asterisk (*).

Entering **@*2,0,0** specifies a point two units in the *X* direction of the last point entered relative to the WCS. Entering **@2,0,0** specifies a point two units in the *X* direction of the last point entered relative to the UCS.

In practice, most coordinates are entered relative to the UCS rather than the WCS.

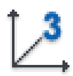
To specify a new UCS with three points

1 Click View tab ➤ UCS panel ➤ 3 Point.

2 Specify a new origin point. This is the point (0,0,0) on the new UCS.

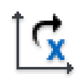
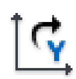
3 Specify a point on the positive *X* axis of the new UCS.

4 Specify a point on the positive *XY* plane of the new UCS.

UCS 

To rotate the UCS around a major axis

1 Do one of the following:

- Click View tab ➤ UCS panel ➤ Z. 
- Click View tab ➤ UCS panel ➤ X. 
- Click View tab ➤ UCS panel ➤ Y. 

2 Enter a rotation angle. A common rotation angle is 90.

Toolbar: UCS

Quick Reference

Commands

UCS

Manages user coordinate systems.

UCSICON

Controls the visibility and placement of the UCS icon.

UCSMAN

Manages defined user coordinate systems.

System Variables

UCSICON

Displays the UCS icon for the current viewport or layout.

UCSVP

Determines whether the UCS in viewports remains fixed or changes to reflect the UCS of the current viewport.

Utilities

No entries

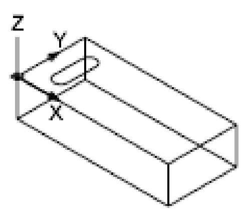
Command Modifiers

No entries

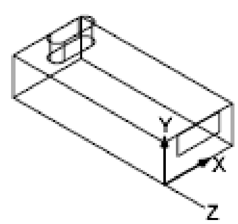
Control the User Coordinate System in 3D

Several methods are available for manipulating the user coordinate system in 3D. You can also save and restore user coordinate system orientations.

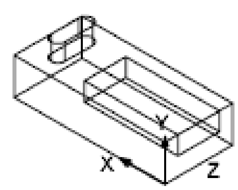
You define a user coordinate system (UCS) to change the location of the 0,0,0 origin point, the location and rotation of the *XY* plane, and the orientation of the *XY* plane or *Z* axis. You can locate and orient a UCS anywhere in 3D space, and you can define, save, and recall as many saved UCS locations as you require.



Moved UCS



Rotated UCS



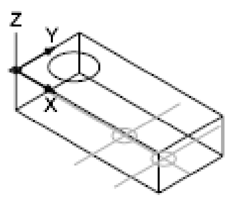
Roriented Z-axis

If multiple viewports are active, you can assign a different UCS to each viewport. With the UCSVP system variable turned on, you can lock a UCS to a viewport, automatically restoring the UCS each time that viewport is made current.

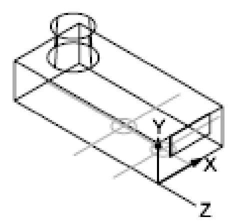
Define the UCS Location

You can define a UCS in the following ways:

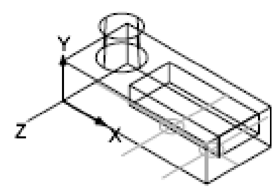
- Specify a new origin (one point), new *X* axis (two points), or new *XY* plane (three points).



UCS origin

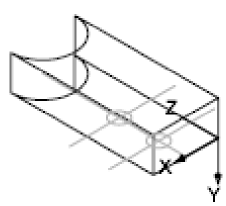


UCS x-axis option

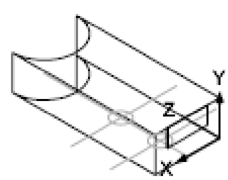


UCS 3-point option

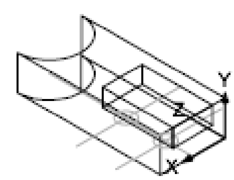
- Align the UCS by selecting a face on a 3D solid object. The selection can be on a face or on an edge of the solid.



UCS face option

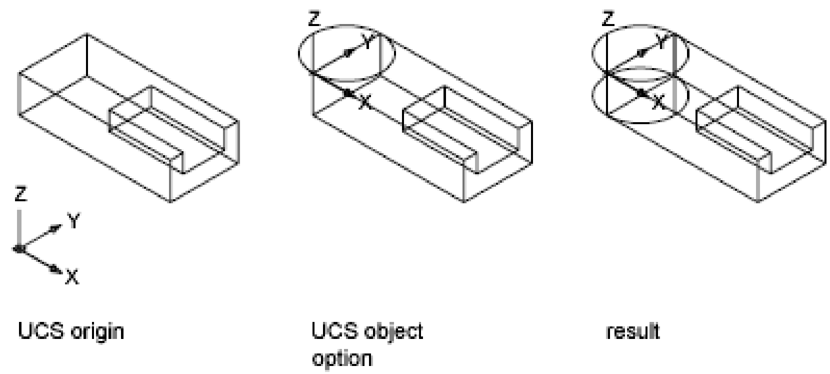


UCS x-flip option

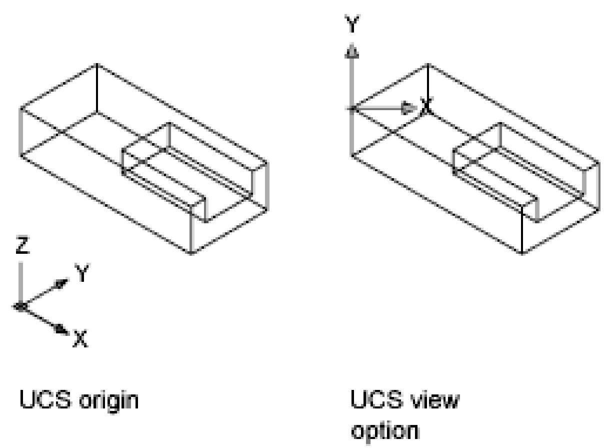


Result

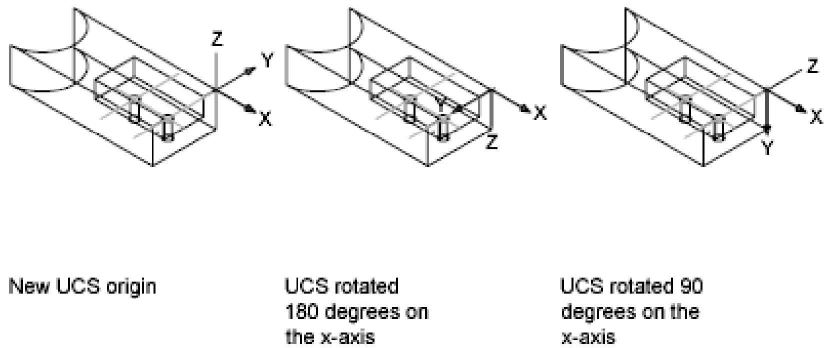
- Align the new UCS with an existing object. The origin of the UCS is located at the vertex nearest to where the object was selected.



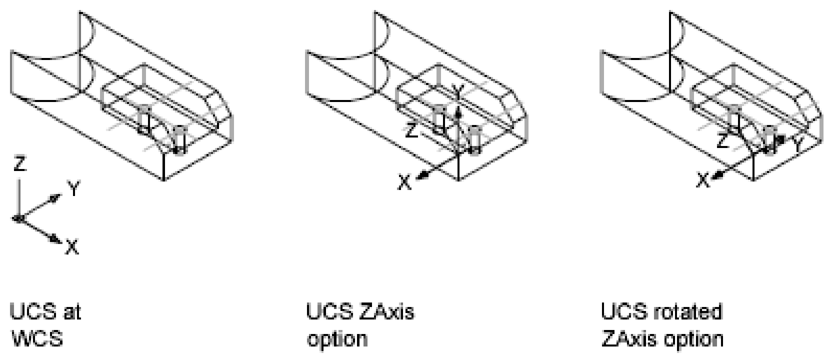
- Align the new UCS with the current viewing direction.



- Rotate the current UCS around any of its three major axes.



- Reorient the *XY* plane by specifying a new *Z* axis.



Use UCS Presets

If you do not want to define your own UCS, you can choose from several preset coordinate systems. The images on the Orthographic UCSs tab of the (Named) UCS dialog box show the available choices.

Change the Default Elevation

The ELEV command sets the default *Z* value for new objects above or below the *XY* plane of the current UCS. This value is stored in the ELEVATION system variable.

NOTE Generally, it is recommended that you leave the elevation set to zero and control the *XY* plane of the current UCS with the UCS command.

Change the UCS in Paper Space

You can define a new UCS in paper space just as you can in model space; however, the UCS in paper space is restricted to 2D manipulation. Although you can enter 3D coordinates in paper space, you cannot use 3D viewing commands such as PLAN and VPOINT.

Save and Restore UCS Locations by Name

If you plan to work extensively in 3D, you can save named UCS locations, each having a different origin and orientation, for various construction requirements. You can relocate, save, and recall as many UCS orientations as you require.

To define a new UCS origin in 3D

1. Click View tab ➤ UCS panel ➤ Origin.
2. Specify a point for the new origin.

   The UCS origin (0,0,0) is redefined at the point you specify.

Toolbar: UCS
Command entry: UCS

To define a new UCS with a specified Z axis

1. Click View tab ➤ UCS panel ➤ Z.
2. Specify a point for the new origin.

   The UCS origin (0,0,0) is redefined at the point you specify.
3. Specify a point that lies on the positive *Z* axis.

Toolbar: UCS
Command entry: UCS

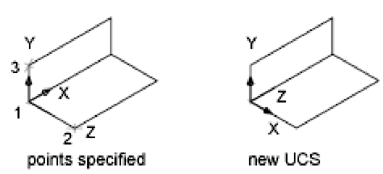
To define a new UCS with particular *X* and *Y* axes

1. Click View tab ➤ UCS panel ➤ 3 Point.
2. Specify a point for the new origin.
   The UCS origin (0,0,0) is redefined at the point you specify.
3. Specify a point that lies on the positive *X* axis.
4. Specify a point that lies on the positive *Y* axis.

Toolbar: UCS
Command entry: UCS

To shift the *XY* plane

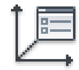
1. Click View tab ➤ UCS panel ➤ 3 Point.
2. Specify an origin point for the new UCS (1).
   For example, in a large drawing, you might specify an origin point near the area in which you want to work.
3. Specify a point to indicate the horizontal orientation of the new UCS (2). This point should be on the positive portion of the new *X* axis.
4. Specify a point to indicate the vertical orientation of the new UCS (3). This point should be on the positive portion of the new *Y* axis.
   The UCS, including grid, shifts to represent the *X* and *Y* axes you have specified.



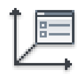
Toolbar: UCS

Command entry: UCS

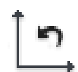
To select a UCS preset

1. Click View tab ➤ UCS panel ➤ Named UCS. 
2. In the UCS dialog box, Orthographic UCSs tab, select a UCS orientation from the list.
3. Click Set Current.
4. Click OK.

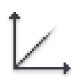
   The UCS changes to the selected option.

Toolbar: UCS II 
Command entry: UCSMAN

To restore the location and orientation of the previous UCS

- Click View tab ➤ UCS panel ➤ Previous. 

The previous UCS is restored.

Toolbar: UCS 
Command entry: UCS

Quick Reference

Commands

ELEV

Sets elevation and extrusion thickness of new objects.

UCS

Manages user coordinate systems.

UCSMAN

Manages defined user coordinate systems.

System Variables

ELEVATION

Stores the current elevation of new objects relative to the current UCS.

UCSVP

Determines whether the UCS in viewports remains fixed or changes to reflect the UCS of the current viewport.

Utilities

No entries

Command Modifiers

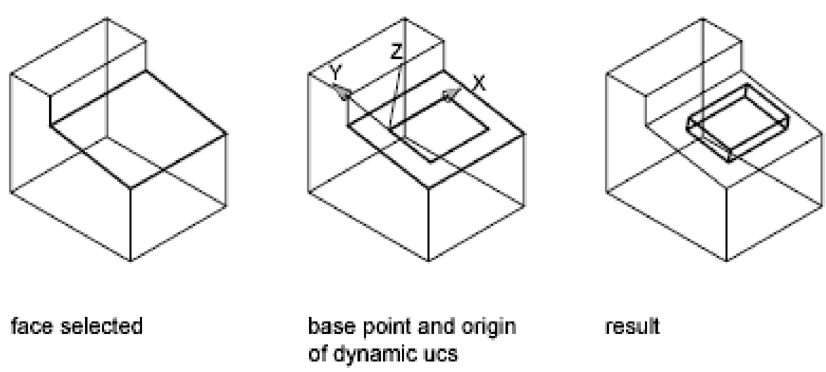
No entries

Use the Dynamic UCS with Solid Models

With the dynamic UCS feature, you can temporarily and automatically align the XY plane of the UCS with a plane on a solid model while creating objects.

When in a draw command, you align the UCS by moving your pointer over an edge of a face rather than having to use the UCS command. After you finish the command, the UCS returns to its previous location and orientation.

For example, you can use the dynamic UCS to create a rectangle on an angled face of a solid model as shown in the illustration.



In the illustration on the left, the UCS is not aligned with the angled face. Instead of relocating the UCS, you turn on the dynamic UCS on the status bar or by pressing F6.

When you move the pointer completely over an edge as shown in the middle illustration, the cursor changes to show the direction of the dynamic UCS axes. You can then create objects on the angled face easily as shown in the illustration on the right.

NOTE To display the *XYZ* labels on the cursor, right-click the DUCS button and click Display Crosshair Labels.

The *X* axis of the dynamic UCS is located along an edge of the face and the positive direction of the *X* axis always points toward the right half of the screen. Only the front faces of a solid are detected by the dynamic UCS.

The types of commands that can use a dynamic UCS include the following:

- **Simple geometry.** Line, polyline, rectangle, arc, circle
- **Text.** Text, Multiline text, table
- **References.** Insert, xref
- **Solids.** Primitives and POLYSOLID
- **Editing.** Rotate, mirror, align
- **Other.** UCS, area, grip tool manipulation

TIP You can easily align the UCS with a plane on a solid model by turning on the dynamic UCS feature and then using the UCS command to locate the origin on that plane.

If Grid and Snap mode are turned on, they align temporarily to the dynamic UCS. The limits of the grid display are set automatically.

You can temporarily turn off the dynamic UCS by pressing F6 or SHIFT+Z while moving the pointer over a face.

NOTE The dynamic UCS is available only while a command is active.

To change the UCS dynamically

1. Start a command supported by dynamic UCS.
2. If necessary, click DUCS on the status bar to turn it on.
3. Move your pointer over the edge of a face on a solid model.
4. Complete the command.

Quick Reference

Commands

OPTIONS

Customizes the program settings.

System Variables

OSOPTIONS

Automatically suppresses object snaps on hatch objects and geometry with negative Z values when using a dynamic UCS.

UCSDETECT

Controls whether dynamic UCS acquisition is active or not.

Utilities

No entries

Command Modifiers

No entries

Assign User Coordinate System Orientations to Viewports

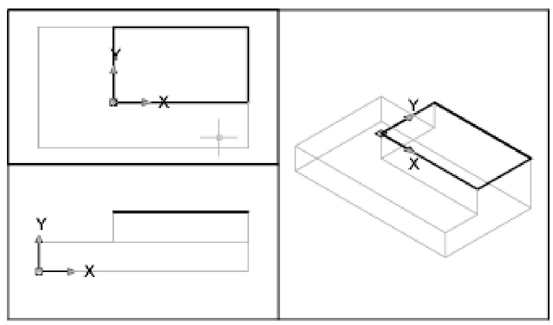
To facilitate editing objects in different views, you can define a different user coordinate system orientation for each view.

Multiple viewports provide different views of your model. For example, you might set up viewports that display top, front, right side, and isometric views. To facilitate editing objects in different views, you can define a different UCS for each view. Each time you make a viewport current, you can begin drawing using the same UCS you used the last time that viewport was current.

The UCS in each viewport is controlled by the UCSVP system variable. When UCSVP is set to 1 in a viewport, the UCS last used in that viewport is saved with the viewport and is restored when the viewport is made current again. When UCSVP is set to 0 in a viewport, its UCS is always the same as the UCS in the current viewport.

For example, you might set up three viewports: a top view, front view, and isometric view. If you set the UCSVP system variable to 0 in the isometric viewport, you can use the Top UCS in both the top viewport and the isometric viewport. When you make the top viewport current, the isometric viewport's UCS reflects the UCS top viewport. Likewise, making the front viewport current switches the isometric viewport's UCS to match that of the front viewport.

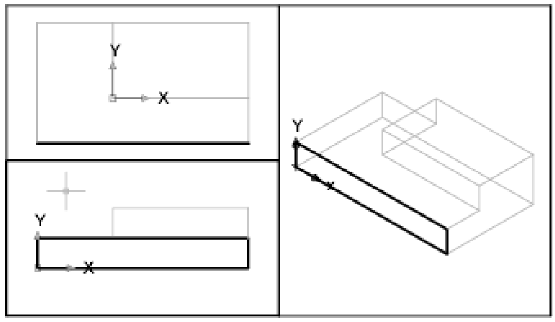
The example is illustrated in the following figures. The first figure shows the isometric viewport reflecting the UCS of the upper-left, or top, viewport, which is current.



top viewport is current

UCSVP = 0; UCS icon reflects UCS of current viewport

The second figure shows the change that occurs when the lower-left, or front, viewport is made current. The UCS in the isometric viewport is updated to reflect the UCS of the front viewport.



front viewport is current

UCSVP = 0; UCS icon reflects UCS of current viewport

In previous releases, the UCS was a global setting for all viewports in either model or paper space. If you want to restore the behavior of previous releases, you can set the value of the UCSVP system variable to 0 in all active viewports.

Quick Reference

Commands

UCS

Manages user coordinate systems.

System Variables

UCSVP

Determines whether the UCS in viewports remains fixed or changes to reflect the UCS of the current viewport.

Utilities

No entries

Command Modifiers

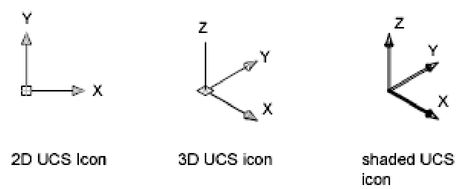
No entries

Control the Display of the User Coordinate System Icon

To help visualize the current orientation of the user coordinate system, you can display the user coordinate system icon. Several versions of this icon are available, and you can change its size, location, and color.

To indicate the location and orientation of the UCS, the UCS icon is displayed either at the UCS origin point or in the lower-left corner of the current viewport.

You can choose one of three styles of icon to represent the UCS.

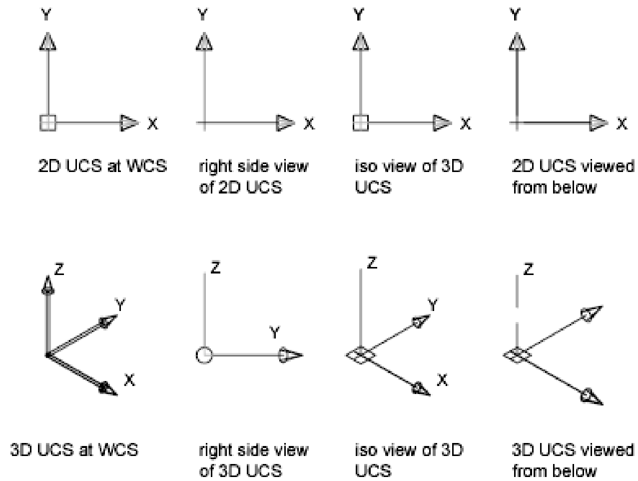


Use the UCSICON command to choose between displaying the 2D or the 3D UCS icon. The shaded UCS icon is displayed for a shaded 3D view. To indicate the origin and orientation of the UCS, you can display the UCS icon at the UCS origin point using the UCSICON command.

If the icon is displayed at the origin of the current UCS, a cross (+) appears in the icon. If the icon is displayed in the lower-left corner of the viewport, no cross appears in the icon.

If you have multiple viewports, each viewport displays its own UCS icon.

The UCS icon is displayed in various ways to help you visualize the orientation of the workplane. The following figure shows some of the possible icon displays.



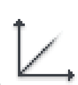
You can use the UCSICON command to switch between the 2D UCS icon and the 3D UCS icon. You can also use the command to change the size, color, arrowhead type, and icon line width of the 3D UCS icon.

The UCS broken pencil icon replaces the 2D UCS icon when the viewing direction is in a plane parallel to the UCS *XY* plane. The broken pencil icon indicates that the edge of the *XY* plane is almost perpendicular to your viewing direction. This icon warns you not to use your pointing device to specify coordinates.

When you use the pointing device to locate a point, it's normally placed on 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—it may be difficult to visualize where the point will be located. In this case, the point will be located on a plane parallel to your viewing plane that also contains the UCS origin point. For example, if the viewing direction is along the *X* axis, coordinates specified with a pointing device will be located on the *YZ* plane, which contains the UCS origin point.

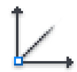
Use the 3D UCS icon to help you visualize which plane these coordinates will be projected on; the 3D UCS icon does not use a broken pencil icon.

To turn the display of the UCS icon on and off

- Click View tab ➤ UCS panel ➤ Toggle Icon. 
  The check mark indicates whether the icon is on or off.

Command entry: UCSICON

To display the UCS icon at the UCS origin

- Click View tab ➤ UCS panel ➤ Origin. 
  The UCS icon is displayed at the origin of the current coordinate system. The check mark indicates whether the option is on or off.

Command entry: UCSICON

To change the appearance of the UCS icon

1. Click View menu ➤ Display ➤ UCS Icon ➤ Properties.
2. In the UCS Icon dialog box, change the settings.
3. Click OK.

Command entry: UCSICON

Quick Reference

Commands

UCSICON

Controls the visibility and placement of the UCS icon.

System Variables

UCSICON

Displays the UCS icon for the current viewport or layout.

Utilities

No entries

Command Modifiers

No entries

Use Dynamic Input

Dynamic Input provides a command interface near the cursor to help you keep your focus in the drafting area.

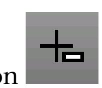
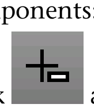
When dynamic input is on, tooltips display information near the cursor that is dynamically updated as the cursor moves. When a command is active, the tooltips provide a place for user entry.

After you type a value in an input field and press TAB, the field then displays a lock icon, and the cursor is constrained by the value that you entered. You can then enter a value for the second input field. Alternately, if you type a value and press ENTER, the second input field is ignored and the value is interpreted as direct distance entry.

The actions required to complete a command or to use grips are similar to those for the command prompt. The difference is that your attention can stay near the cursor.

Dynamic input is not designed to replace the command window. You can hide the command window to add screen area for drawing, but you will need to display it for some operations. Press F2 to hide and display command prompts and error messages as needed. Alternately, you can undock the command window and use Auto-hide to roll open or roll up the window.

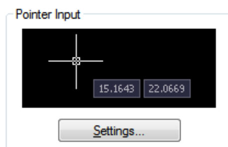
Turn On or Turn Off Dynamic Input

Click the dynamic input button  on the status bar to turn dynamic input on and off. You can turn it off temporarily by holding down the F12 key. Dynamic input has three components: pointer input, dimensional input, and dynamic prompts. Right-click  and click Settings to control what is displayed by each component when dynamic input is on.

Pointer Input

When pointer input is on and a command is active, the location of the crosshairs is displayed as coordinates in a tooltip near the cursor. You can enter coordinate values in the tooltip instead of on the command line.

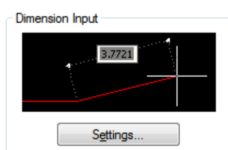
The default for second and subsequent points is relative polar coordinates (relative Cartesian for RECTANG). There is no need to type the at sign (@). If you want to use absolute coordinates, use the pound sign (#) prefix. For example, to move an object to the origin, for the second point prompt, enter **#0,0**.



Use the pointer input settings to change the default format for coordinates and to control when pointer input tooltips are displayed.

Dimensional Input

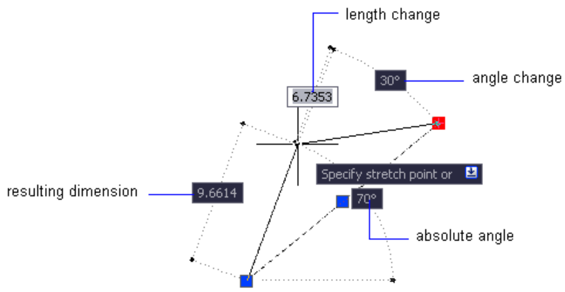
When dimensional input is on, the tooltips display distance and angle values when a command prompts for a second point. The values in the dimensional tooltips change as you move the cursor. Press TAB to move to the value you want to change. Dimensional input is available for ARC, CIRCLE, ELLIPSE, LINE, and PLINE.



When you use grips to edit an object, the dimensional input tooltips can display the following information:

- Original length
- A length that updates as you move the grip
- The change in the length
- Angle
- The change in the angle as you move the grip

■ The radius of an arc

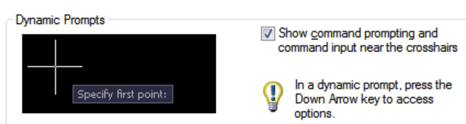


Use the dimensional input settings to display only the information you want to see.

When you use grips to stretch objects or when you create new objects, dimensional input displays only acute angles, that is, all angles are displayed as 180 degrees or less. Thus, an angle of 270 degrees is displayed as 90 degrees regardless of the ANGDIR system variable setting (set in the Drawing Units dialog box). Angles specified when creating new objects rely on the cursor location to determine the positive angle direction.

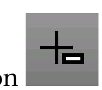
Dynamic Prompts

When dynamic prompts are on, prompts are displayed in a tooltip near the cursor. You can enter a response in the tooltip instead of on the command line. Press the DOWN ARROW key to view and select options. Press the UP ARROW key to display recent input.



NOTE To use PASTECLIP in a dynamic prompt tooltip, type a letter and then backspace to delete it before you paste the entry. Otherwise, the entry is pasted into the drawing as text.

To enter coordinate values in dynamic input tooltips

1 On the status bar, verify that the dynamic input button  is on.

2 Use one of the following methods to enter coordinate values or select options:
   - To enter polar coordinates, enter the distance from the first point and press TAB, and then enter an angle value and press ENTER.
   - To enter Cartesian coordinates, enter an *X* coordinate value and a comma (,), and then enter a *Y* coordinate value and press ENTER.
   - If a down-arrow icon follows the prompt, press the DOWN ARROW key until a dot is displayed next to the option. Press ENTER.
   - Press the UP ARROW key to access recent coordinates, or right-click and click Recent Input to access the coordinates from a shortcut menu.

NOTE For dimensional input, after you type a value in an input field and press TAB, the field then displays a lock icon, and the cursor is constrained by the value that you entered.

To correct typing errors in dynamic input tooltips

- When a dynamic input tooltip displays the red error outline, the current entry is selected. Type over the selected text to replace it. You can also use the RIGHT ARROW, LEFT ARROW, BACKSPACE, and DELETE keys to correct your entry. After you make the correction, press TAB, comma (,), or a left angle bracket (<) to remove the red outline and complete the coordinates.
- If you type the @ or # or * prefixes in a pointer input tooltip and then want to change it, you can just type the character you want. There is no need to backspace.

To specify relative or absolute coordinates in pointer input tooltips

- To enter absolute coordinates when relative coordinates are displayed in the tooltip, enter # to temporarily override the DYNPICOORDS system variable.
- To enter relative coordinates when absolute coordinates are displayed, enter @ to temporarily override the DYNPICOORDS system variable.
- To enter absolute world coordinate system (WCS) coordinates, enter *.

NOTE During pointer input, you can use the shortcut menu to access the # and * prefixes.

To choose options in dynamic prompt tooltips

1 Start a command.

   Tooltips near the crosshairs display the coordinates of the cursor location and a prompt.

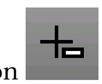
2 If a prompt is displayed, press the DOWN ARROW key to display the options.

3 Press DOWN ARROW or UP ARROW to place a dot next to the option you want. Press ENTER.

   Press ESC to undo the most recent action.

4 Specify points and enter options to complete the command.

To change the color, size, or transparency of tooltips

1 Right-click the dynamic input button  on the status bar. Click Settings.

2 In the Drafting Settings dialog box, Dynamic Input tab, click Drafting Tooltip Appearance.

3 In the Tooltip Appearance dialog box, under Color, click Model Color or Layout Color to display the Select Color dialog box, where you can specify a color for tooltips in the space you selected.

4 Under Size, move the slider to the right to make tooltips larger or to the left to make them smaller. The default value, 0, is in the middle.

5 Under Transparency, move the slider. The lower the setting, the more transparent the tooltip. A value of 100 sets the tooltip to opaque.

6 Under Apply To, choose an option:

   - **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.

7 Click OK.

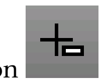
Command entry: DYNTOOLTIPS

To merge the information in all drafting tooltips into one tooltip

- At the command prompt, enter **tooltipmerge**.

Tooltips that display drafting information are merged into one.

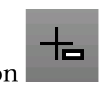
To turn on or turn off dynamic input

- On the status bar, click the dynamic input button  , or press F12.

To turn off dynamic input temporarily, hold down the F12 key while you work.

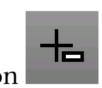
NOTE The F12 temporary override key does not turn on dynamic input.

To change dynamic input settings

1. Right-click the dynamic input button  on the status bar. Click Settings.
2. Change the dynamic input settings as needed.
3. Click OK to close each dialog box.

Command entry: DSETTINGS

To change pointer input settings

1. Right-click the dynamic input button  on the status bar. Click Settings.
2. In the Drafting Settings dialog box, Dynamic Input tab, under Pointer Input, click Settings.
3. In the Pointer Input Settings dialog box, select polar or Cartesian format as the default.
4. Select relative or absolute coordinate format as the default.

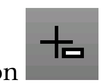
5 Under Visibility, select one of the following options:

- **As Soon As I Type Coordinate Data**. When pointer input is turned on, displays tooltips only when you start to enter coordinate data.
- **When a Command Asks for a Point**. When pointer input is turned on, displays tooltips whenever a command prompts for a point.
- **Always—Even When Not in a Command**. Always displays tooltips when pointer input is turned on.

6 Click OK to close each dialog box.

Command entry: DSETTINGS

To change dimensional input settings

1 Right-click the dynamic input button  on the status bar. Click Settings.

2 In the Drafting Settings dialog box, Dynamic Input tab, under Dimension Input, click Settings.

3 In the Dimension Input Settings dialog box, select Polar or Cartesian format as the default.

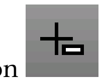
4 Under Visibility, select one of the following options:

- **Show Only 1 Dimension Input Field at a Time**. Displays only the distance dimensional input tooltip when you are using grip editing to stretch an object.
- **Show 2 Dimension Input Fields at a Time**. Displays the distance and angle dimensional input tooltips when you are using grip editing to stretch an object.
- **Show the Following Dimension Input Fields Simultaneously**. Displays the selected dimensional input tooltips when you are using grip editing to stretch an object. Select one or more of the check boxes.

5 Click OK to close each dialog box.

Command entry: DSETTINGS

To display prompts in tooltips

1. Right-click the dynamic input button  on the status bar. Click Settings.
2. In the Drafting Settings dialog box, Dynamic Input tab, under Dynamic Prompts, check Show Command Prompting and Command Input Near the Crosshairs.
3. Click OK.

Command entry: DSETTINGS

Quick Reference

Commands

DSETTINGS

Sets grid and snap, polar and object snap tracking, object snap modes, Dynamic Input, and Quick Properties.

System Variables

DYNDIGRIP

Controls which dynamic dimensions are displayed during grip stretch editing.

DYNDIVIS

Controls how many dynamic dimensions are displayed during grip stretch editing.

DYNMODE

Turns Dynamic Input features on and off.

DYNPICOORDS

Controls whether pointer input uses relative or absolute format for coordinates.

DYNPIFORMAT

Controls whether pointer input uses polar or Cartesian format for coordinates.

DYNPIVIS

Controls when pointer input is displayed.

DYNPROMPT

Controls display of prompts in Dynamic Input tooltips.

DYNTOOLTIPS

Controls which tooltips are affected by tooltip appearance settings.

TEMPOVERRIDES

Turns temporary override keys on and off.

TOOLTIPMERGE

Combines drafting tooltips into a single tooltip.

Utilities

No entries

Command Modifiers

No entries

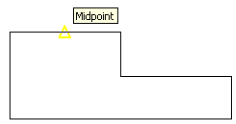
Snap to Locations on Objects (Object Snaps)

Instead of entering coordinates, you can specify points relative to existing objects such as endpoints of lines or center points of circles.

Use Object Snaps

Use object snaps to specify precise locations on objects. For example, you can use an object snap to draw a line to the center of a circle or to the midpoint of a polyline segment.

You can specify an object snap whenever you are prompted for a point. By default, a marker and a tooltip are displayed when you move the cursor over an object snap location on an object. This feature, called AutoSnap™, provides a visual clue that indicates which object snaps are in effect.



For a list of object snaps, see OSNAP.

Specify an Object Snap

To specify an object snap at a prompt for a point, you can

- Press SHIFT and right-click to display the Object Snap shortcut menu
- Click an object snap button on the Object Snap toolbar
- Enter the name of an object snap at the command prompt
- On the status bar, right-click the object snap button

When you specify an object snap at a prompt for a point, the object snap stays in effect only for the next point that you specify.

NOTE Object snaps work only when you are prompted for a point. If you try to use an object snap at the command prompt, an error message is displayed.

Use Running Object Snaps

If you need to use one or more object snaps repeatedly, you can turn on *running object snaps*. For example, you might set Center as a running object snap if you need to connect the centers of a series of circles with a line.

You can specify one or more running object snaps on the Object Snaps tab in the Drafting Settings dialog box, which is accessible from the Tools menu. If several running object snaps are on, more than one object snap may be eligible at a given location. Press TAB to cycle through the possibilities before you specify the point.

Click the OSNAP button on the status bar or press F3 to turn running object snaps on and off.

NOTE If you want object snaps to ignore hatch objects, set the OSOPTIONS system variable to 1.

Use Object Snaps in 3D

By default, the *Z*-value of an object snap location is determined by the object's location in space. However, if you work with object snaps on the plan view of a building or the top view of a part, a constant *Z*-value is more useful.

If you turn on the OSNAPZ system variable, all object snaps are projected onto the *XY* plane of the current UCS or, if ELEV is set to a non-zero value, onto a plane parallel to *XY* plane at the specified elevation.

NOTE When you draw or modify objects, make sure that you know whether OSNAPZ is on or off. There is no visual reminder, and you can get unexpected results.

To snap to a geometric point on an object

1. At the prompt for a point, hold down SHIFT and right-click in the drawing area. Select the object snap you want to use.
2. Move your cursor over the desired object snap location.

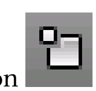
   If AutoSnap is on, your cursor automatically locks onto the snap location you selected, and a marker and tooltip indicate the object snap point.
3. Select an object.

   The cursor snaps to the eligible location closest to your selection.

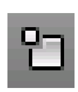
To set running object snaps

1. Click Tools menu ➤ Drafting Settings.
2. In the Drafting Settings dialog box, Object Snap tab, select the object snaps you want to use.
3. Click OK.

Command entry: OSNAP

Shortcut menu: Right-click the OSNAP button  on the status bar. Click Settings.

To turn on and turn off running object snaps as you work

- On the status bar, click Osnap , or press F3.

If running object snaps have been set, the settings are turned on or off.

To turn running object snaps on and off temporarily, hold down the F3 key while you work.

Command entry: OSNAP

To set object snaps to ignore hatch objects

1. At the command prompt, enter **osoptions**.
2. Enter **1**.

Quick Reference

Commands

APERTURE

Sets the display size for the object snap target box, in pixels.

OPTIONS

Customizes the program settings.

OSNAP

Sets running object snap modes.

System Variables

APBOX

Turns the display of the AutoSnap aperture box on or off.

AUTOSNAP

Controls the display of the AutoSnap marker, tooltip, and magnet.

OSMODE

Sets running object snaps

OSNAPZ

Controls whether object snaps are automatically projected onto a plane parallel to the XY plane of the current UCS at the current elevation.

OSNAPCOORD

Controls whether coordinates entered on the command line will override running object snaps.

OSOPTIONS

Automatically suppresses object snaps on hatch objects and geometry with negative Z values when using a dynamic UCS.

Utilities

No entries

Command Modifiers

MTP (Command Modifier)

Locates the midpoint between two points.

The Object Snap Menu

Specify an object snap quickly and conveniently from a shortcut menu.

The object snap menu is displayed at your cursor location when you hold down SHIFT and click the right mouse button or the equivalent button on another pointing device.

The default object snap menu lists object snaps and tracking options. If you want to change the options, you can modify a customization file. The main customization file that's shipped with the product is *acad.cuix*.

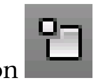
See also:

- "Pull-down and Shortcut Menus" in the *Customization Guide*
- Use Object Snaps on page 676

To display the object snap menu

1. Enter any command that prompts you to specify a point. For example, enter **line**.
2. At the From Point prompt, hold down SHIFT and right-click.

   The object snap menu is displayed, and you can click an object snap option.

Shortcut menu: On the status bar, right-click the object snap button  . Click Settings.

Quick Reference

Commands

OSNAP

Sets running object snap modes.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Set Visual Aids for Object Snaps (AutoSnap)

Object snaps include a visual aid called AutoSnap™ to help you see and use object snaps more efficiently. AutoSnap displays a marker and a tooltip when you move your cursor over an object snap location.

AutoSnap Tools

AutoSnap consists of the following snap tools:

- **Marker.** Displays the object snap location when the cursor moves over or near an object. Marker shape is dependent on the snap it is marking.
- **Tooltip.** Describes which part of the object you are snapping to in a small flag at the cursor location.
- **Magnet.** Attracts and locks the cursor onto the nearest detected snap points. Provides a visual cue, similar to snapping to a grid.

■ **Aperture box.** Surrounds the crosshairs and defines an area within which object snaps are evaluated. You can choose to display or not display the aperture box, and you can change the aperture box size.

The AutoSnap markers, tooltips, and magnet are turned on by default. You can change AutoSnap settings on the Drafting tab in the Options dialog box.

Use AutoSnap to Confirm or Change an Object Snap

If you have set more than one running object snap, you can press TAB to cycle through all the object snap points available for a particular object.

To change the AutoSnap settings

1 Click Tools menu ➤ Options. 

2 In the Options dialog box, Drafting tab, change settings as needed:
   - **Marker.** Turns the marker on or off.
   - **Magnet.** Turns the magnet on or off.
   - **Display AutoSnap Tooltip.** Turns tooltips on or off.
   - **Display AutoSnap Aperture Box.** Turns the target box on or off when you specify an object snap. This setting does not affect object snaps when you are not using AutoSnap.
   - **AutoSnap Marker Color.** Changes the color of the marker.
   - **AutoSnap Marker Size.** Adjusts the size of the marker.

3 Click OK.

Command entry: OPTIONS

Quick Reference

Commands

APERTURE

Sets the display size for the object snap target box, in pixels.

OPTIONS

Customizes the program settings.

OSNAP

Sets running object snap modes.

System Variables

APBOX

Turns the display of the AutoSnap aperture box on or off.

AUTOSNAP

Controls the display of the AutoSnap marker, tooltip, and magnet.

OSMODE

Sets running object snaps

Utilities

No entries

Command Modifiers

No entries

Override Object Snap Settings

While you work, you can turn running object snaps on and off temporarily by using an override key. Temporary override keys can also be used for other drawing aids; for example, Ortho mode and Polar mode.

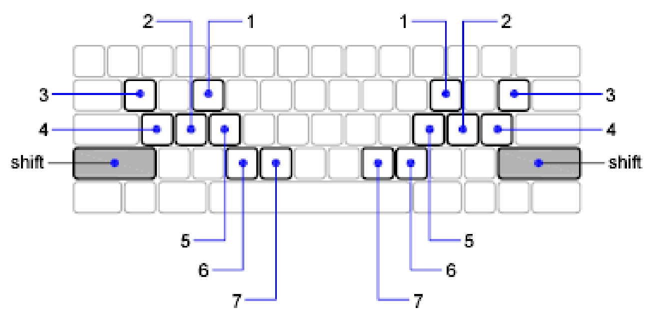
For example, if you have set running object snaps but you want to turn them off for one point, you can hold down F3. When you release this override key, running object snaps are restored.

There are also temporary override keys for individual object snaps. Override keys are set up to be easy to find by touch without looking away from your drawing.

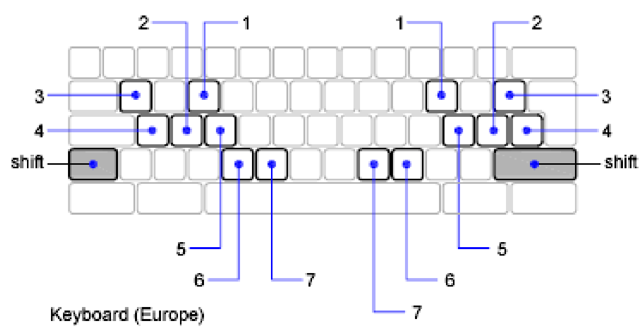
The keys in the following illustration are the default keys, but you can change key assignments and add your own as needed.

Hold down SHIFT and one of the temporary override keys in the illustration:

1	Object snap override: Endpoint	5	Turns off all snapping and tracking
2	Enforces object snap selection	6	Object snap override: Center
3	Toggles object snap tracking	7	Object snap override: Midpoint
4	Toggles object snap mode (OSNAP)		



Keyboard (U.S. and Japan)



Keyboard (Europe)

Temporary override keys are also available for the other drawing aids that you set in the Drafting Settings dialog box.

See also:

- "Adjust Grid and Grid Snap"
- "Use Orthogonal Locking (Ortho Mode)"
- "Use Polar Tracking and PolarSnap"
- "Use Dynamic Input"

- Keyboard Shortcuts in the *Customization Guide*

To temporarily override the running object snap settings

- Hold down F3 while you work.
  When you release the key, the current running object snap settings are restored.

To change the keyboard response time for temporary override keys

1. In the Windows Control Panel, click Keyboard.
2. In the Keyboard Properties dialog box, on the Speed tab, drag the Repeat Rate slider to adjust the keyboard response time. Then click OK.

Quick Reference

Commands

CUI

Manages the customized user interface elements in the product.

CUSTOMIZE

Customizes tool palettes and tool palette groups.

OPTIONS

Customizes the program settings.

OSNAP

Sets running object snap modes.

System Variables

OSMODE

Sets running object snaps

TEMPOVERRIDES

Turns temporary override keys on and off.

Utilities

No entries

Command Modifiers

No entries

Restrict Cursor Movement

Several tools are available that you can use to restrict or lock the movement of your cursor.

Adjust Grid and Grid Snap

To enhance drawing speed and efficiency, you can display and snap to a rectangular grid. You can also control its spacing, angle, and alignment.

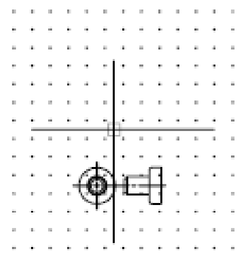
The grid is a rectangular pattern of dots or lines that extends over the area you specify as the grid limits. Using the grid is similar to placing a sheet of grid paper under a drawing. The grid helps you align objects and visualize the distances between them. The grid is not plotted.

Snap mode restricts the movement of the crosshairs to intervals that you define. When Snap mode is on, the cursor seems to adhere, or "snap," to an invisible rectangular grid. Snap is useful for specifying precise points with the arrow keys or the pointing device.

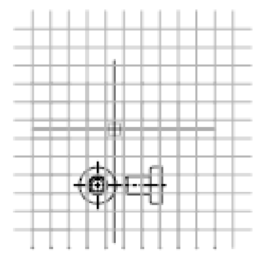
Grid mode and Snap mode are independent but are often turned on at the same time.

Control the Display Style and Area of the Grid

You can display the grid either as a rectangular pattern of dots or as rectangular pattern of lines. The grid displays dots only when the current visual style is set to 2D Wireframe, otherwise the grid displays lines. A lined grid is displayed for all visual styles while working in 3D. There are several methods to change the current visual style, including the VSCURRENT command.



Visual styles = 2D wireframe



Visual styles = hidden

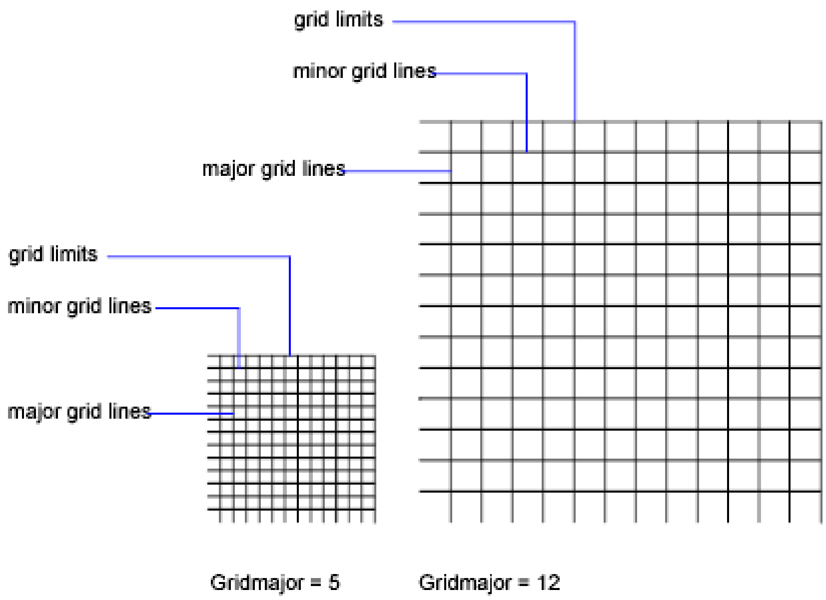
By default, the *X* and *Y* axes of the UCS display in a different color than the grid lines. You can control the color in the Drawing Window Colors dialog box. This dialog box is accessible from the Drafting tab in the Options dialog box.

The LIMITS command controls the drawing area covered by the grid. As an option, you can override the limits to make the grid cover the entire *XY* plane of the user coordinate system (UCS). You can access this option in the Drafting Settings dialog box or use the GRIDDISPLAY system variable.

NOTE When you use dynamic UCS, the grid limits are set automatically relative to the size of the selected face of the solid and the drawing area available.

Control the Frequency of Major Grid Lines

If the grid is displayed as lines rather than dots, darker lines called *major grid lines* display at intervals. When working in decimal units or with feet and inches, major grid lines are especially useful for measuring distances quickly. You can control the frequency of major grid lines in the Drafting Settings dialog box.



To turn off the display of major grid lines, set the frequency of major grid lines to 1.

NOTE If the grid is displayed as lines, the grid limits are displayed also as darker lines. Do not confuse these boundaries with major grid lines.

NOTE When the grid is displayed as lines and SNAPANG is set to a value other than 0, the grid will not display. SNAPANG does not affect the display of the dotted grid.

Change the Grid Dynamically During Zooming

If you zoom in or out of your drawing, the grid spacing is adjusted automatically to be more appropriate for the new magnification. This is called *adaptive grid display*.

For example, if you zoom way out, the density of displayed grid lines reduces automatically. Conversely, if you zoom way in, additional grid lines display in the same proportion as the major grid lines.

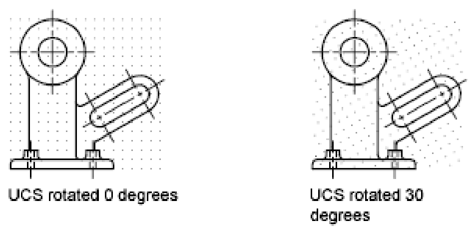
Change Grid and Snap Spacing

As you work, you can turn Grid and Snap mode on and off, and you can change the grid and snap spacing. You can turn Snap mode on and off temporarily by using an override key.

Snap spacing does not have to match grid spacing. For example, you might set a wide grid spacing to be used as a reference but maintain a closer snap spacing for accuracy in specifying points.

Change the Grid and Snap Angle and Base

If you need to draw along a specific alignment or angle, you can change the grid and snap angle by rotating the user coordinate system (UCS). This rotation realigns the crosshairs on the screen to match the new angle. In the following example, the UCS is rotated 30 degrees to match the angle of the anchor bracket.



The grid and snap points are always aligned with the UCS origin. If you need to shift the grid and grid snap origin, move the UCS.

See also:

- "Set Isometric Grid and Snap"
- Override Object Snap Settings on page 683
- Use a Visual Style to Display Your Model on page 293

To display a grid and set grid spacing

1 Click Tools menu ➤ Drafting Settings.

2 In the Drafting Settings dialog box, Snap and Grid tab, select Grid On to display the grid.

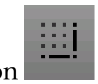
3 Under Snap Type, make sure Grid Snap and Rectangular Snap are selected.

4 For Grid *X* Spacing, enter the horizontal grid spacing in units.

5 To use the same value for vertical grid spacing, press ENTER. Otherwise, enter a new value for Grid *Y* Spacing.

6 Click OK.

Command entry: DSETTINGS

Shortcut menu: On the status bar, right-click the snap button . Click Settings.

To turn on Snap mode and set snap spacing

1 Click Tools menu ➤ Drafting Settings.

2 In the Drafting Settings dialog box, Snap and Grid tab, select Snap On.

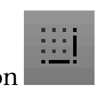
3 Under Snap Type, make sure Grid Snap and Rectangular Snap are selected.

4 In the Snap *X* Spacing box, enter the horizontal snap spacing value in units.

5 To specify the same vertical snap spacing, press ENTER. Otherwise, enter a new distance in the Snap *Y* Spacing box.

6 Click OK.

Command entry: DSETTINGS

Shortcut menu: On the status bar, right-click the snap button . Click Settings.

To set the grid limits

1 Click Format menu ➤ Drawing Limits.

2 At the command prompt, enter the coordinate values for a point at the lower left corner of the grid limits.

3 Enter the coordinate values for a point at the upper right corner of the grid limits.

The grid limits are set to a rectangular area defined by the two points.

Command entry: LIMITS

To temporarily override Snap mode

■ Hold down F9 while you work.
When you release the key, Snap mode is restored.

To rotate the grid and snap angle and change the base point

1 Click View tab ➤ UCS panel ➤ Z.

2 Enter the rotation angle for the UCS.

3 Click View tab ➤ UCS panel ➤ Origin.

4 Specify a new origin point for the UCS.

5 Click OK.

Command entry: UCS

To change the grid display between dots and lines

1 At the command prompt, enter VSCURRENT.

2 Do *one* of the following:

- To display the grid as dots, specify the 2D Wireframe option.
- To display the grid as lines, specify any other option.

Command entry: VSCURRENT

To change the frequency of major grid lines

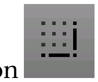
1 If necessary, at the command prompt, enter VSCURRENT and specify any visual style other than 2D Wireframe.

2 Click Tools menu ➤ Drafting Settings.

3 In the Drafting Settings dialog box, Snap and Grid tab, specify a number for Major Line Every.

4 Click OK.

Command entry: VSCURRENT

Shortcut menu: On the status bar, right-click the snap button . Click Settings.

Quick Reference

Commands

DSETTINGS

Sets grid and snap, polar and object snap tracking, object snap modes, Dynamic Input, and Quick Properties.

GRID

Displays a grid pattern in the current viewport.

LIMITS

Sets and controls the limits of the grid display in the current Model or layout tab.

SHADEMODE

Starts the VSCURRENT command.

SNAP

Restricts cursor movement to specified intervals.

System Variables

GRIDDISPLAY

Controls the display behavior and display limits of the grid.

GRIDMODE

Specifies whether the grid is turned on or off.

GRIDMAJOR

Controls the frequency of major grid lines compared to minor grid lines.

GRIDUNIT

Specifies the grid spacing (X and Y) for the current viewport.

LIMCHECK

Controls the creation of objects outside the grid limits.

LIMMAX

Stores the upper-right grid limits for the current space, expressed as world coordinates.

LIMMIN

Stores the lower-left grid limits for the current space, expressed as a world coordinate.

SNAPANG

Sets the snap and grid rotation angle for the current viewport relative to the current UCS.

SNAPBASE

Sets the snap and grid origin point for the current viewport relative to the current UCS.

SNAPMODE

Turns the Snap mode on and off.

SNAPTYPE

Sets the type of snap for the current viewport.

SNAPUNIT

Sets the snap spacing for the current viewport.

TEMPOVERRIDES

Turns temporary override keys on and off.

Utilities

No entries

Command Modifiers

No entries

Use Orthogonal Locking (Ortho Mode)

You can restrict cursor movement to horizontal and vertical for convenience and precision when creating and modifying objects.

As you create or move objects, you can use Ortho mode to restrict the cursor to the horizontal or vertical axis. As you move the cursor, the rubber-band line follows the horizontal or vertical axis, whichever is nearest the cursor.

The orientation of the current user coordinate system (UCS) determines the horizontal and vertical directions. In 3D views, Ortho mode additionally restricts the cursor to the up and down directions. In that case, the tooltip displays a +Z or -Z for the angle.

TIP Use direct distance entry with Ortho mode turned on to create orthogonal lines of specified lengths or to move objects specified distances.

You can turn Ortho on and off at any time during drawing and editing. Ortho is ignored when you enter coordinates or specify an object snap. To turn Ortho on or off temporarily, hold down the temporary override key, SHIFT. While you use the temporary override key, the direct distance entry method is not available.

For drawing or editing objects at angles that are not parallel to the horizontal or vertical axis, see Use Polar Tracking and PolarSnap on page 695.

If turned on, the isometric snap setting takes priority over the UCS in determining horizontal and vertical directions.

NOTE Ortho mode and polar tracking cannot be on at the same time. Turning on Ortho turns off polar tracking.

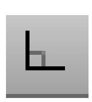
See also:

- Override Object Snap Settings on page 683

To turn on or turn off Ortho mode

- On the status bar, click the Ortho button .
  To turn Ortho on or off temporarily, hold down the SHIFT key while you work. While you use the temporary override key, the direct distance entry method is not available.

  NOTE Turning on Ortho automatically turns off polar tracking.

Command entry: ORTHO

Quick Reference

Commands

ORTHO

Constrains cursor movement to the horizontal or vertical direction.

System Variables

ORTHOMODE

Constrains cursor movement to the perpendicular.

TEMPOVERRIDES

Turns temporary override keys on and off.

Utilities

No entries

Command Modifiers

No entries

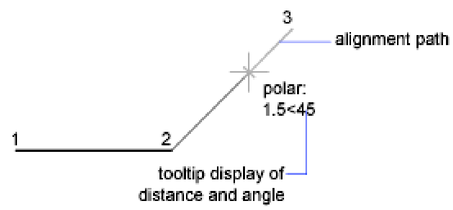
Use Polar Tracking and PolarSnap

Polar tracking restricts cursor movement to specified angles. PolarSnap restricts cursor movement to specified increments along a polar angle.

When you are creating or modifying objects, you can use polar tracking to display temporary alignment paths defined by the polar angles you specify. In 3D views, polar tracking additionally provides an alignment path in the up and down directions. In that case, the tooltip displays a +Z or -Z for the angle.

Polar angles are relative to the orientation of the current user coordinate system (UCS) and the setting for the base angle convention in a drawing. The angle base direction is set in the Drawing Units dialog box.

Use PolarSnap™ to snap to specified distances along the alignment path. For example, in the following illustration you draw a two-unit line from point 1 to point 2, and then draw a two-unit line to point 3 at a 45-degree angle to the line. If you turn on the 45-degree polar angle increment, an alignment path and tooltip are displayed when your cursor crosses the 0 or 45-degree angle. The alignment path and tooltip disappear when you move the cursor away from the angle.

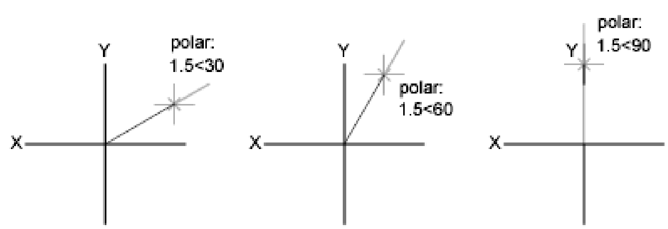


As you move your cursor, alignment paths and tooltips are displayed when you move the cursor near polar angles. The default angle measurement is 90 degrees. Use the alignment path and tooltip to draw your object. You can use polar tracking with Intersection and Apparent Intersection object snaps to find where a polar alignment path intersects another object.

NOTE Ortho mode and polar tracking cannot be on at the same time. Turning on polar tracking turns off Ortho mode. Similarly, PolarSnap and grid snap cannot be on at the same time. Turning on PolarSnap turns off grid snap.

Specify Polar Angles (Polar Tracking)

You can use polar tracking to track along polar angle increments of 90, 60, 45, 30, 22.5, 18, 15, 10, and 5 degrees, or you can specify other angles. The following illustration shows the alignment paths displayed as you move your cursor 90 degrees with the polar angle increment set to 30 degrees.



The orientation of 0 depends on the angle you set in the Drawing Units dialog box (UNITS). The direction of snap (clockwise or counterclockwise) depends on the units direction you specify when setting units of measurement.

You can turn polar tracking on and off temporarily by using an override key. The direct distance entry method is not available while you are using the temporary override key for polar tracking.

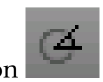
Specify Polar Distances (PolarSnap)

PolarSnap restricts cursor movement to increments of a polar distance you specify. For example, if you specify a length of 4 units, the cursor snaps from the first point specified to lengths of 0, 4, 8, 12, 16, and so on. As you move your cursor, a tooltip indicates the nearest PolarSnap increment. To restrict point entry to polar distances, both polar tracking and Snap mode (set to PolarSnap) must be on. You can turn off all snapping and tracking temporarily by using an override key.

See also:

- Override Object Snap Settings on page 683

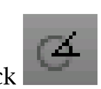
To turn on and turn off polar tracking

- Press F10, or click the polar button  on the status bar.
  To turn polar tracking on or off temporarily, hold down the F10 key while you work.

To set polar snap distance

1. Click Tools menu ➤ Drafting Settings.
2. In the Drafting Settings dialog box, Snap and Grid tab, select Snap On.
3. In Snap Type, select PolarSnap.
4. Under Polar Spacing, enter the polar distance.
5. On the Polar Tracking tab, select Polar Tracking On.
6. Select the angle from the Increment Angle list.
   You can specify your own angles by choosing Additional Angles and then New.
7. Click OK.

Command entry: DSETTINGS

Shortcut menu: On the status bar, right-click .

To draw objects using polar tracking

1. Turn on polar tracking and start a drawing command, such as ARC, CIRCLE, or LINE.

   You can also use polar tracking with editing commands, such as COPY and MOVE.

2. As you move your cursor to specify points, notice the dotted polar tracking line that appears at the tracking angles you specified. Points you specify while the line is displayed conform to the polar tracking angle.

Command entry: DSETTINGS

To draw objects using polar distance

1. Turn on snap and polar tracking.

   Make sure Polar Snap is selected in the Drafting Settings dialog box, Snap & Grid tab.

2. Start a drawing command, such as LINE.
3. As you move your cursor, notice that the dotted polar tracking line displays a tooltip that shows distance and angle.
4. Specify a point.

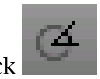
   The length of the new line conforms to the polar distance.

Command entry: DSETTINGS

To set polar tracking angles

1. Click Tools menu ➤ Drafting Settings.
2. In the Drafting Settings dialog box, Polar Tracking tab, select Polar Tracking On.
3. In the Increment Angle list, select the polar tracking angle.
4. To set additional tracking angles, select Additional Angles. Click New. Enter the angle value in the text box.
5. Under Polar Angle Measurement, specify whether polar tracking increments are based on the UCS or relative to the last object you created.
6. Click OK.

Command entry: DSETTINGS

On the status bar, right-click  . Click an available angle or Settings to set additional tracking angles.

Quick Reference

Commands

DSETTINGS

Sets grid and snap, polar and object snap tracking, object snap modes, Dynamic Input, and Quick Properties.

SNAP

Restricts cursor movement to specified intervals.

UNITS

Controls coordinate and angle display formats and precision.

System Variables

ANGBASE

Sets the base angle to 0 with respect to the current UCS.

ANGDIR

Sets the direction of positive angles.

AUTOSNAP

Controls the display of the AutoSnap marker, tooltip, and magnet.

POLARANG

Sets the polar angle increment.

POLARDIST

Sets the snap increment when the SNAPTYPE is set to 1 (PolarSnap).

POLARMODE

Controls settings for polar and object snap tracking.

SNAPTYPE

Sets the type of snap for the current viewport.

TEMPOVERRIDES

Turns temporary override keys on and off.

TRACKPATH

Controls the display of polar and object snap tracking alignment paths.

Utilities

No entries

Command Modifiers

No entries

Lock an Angle for One Point (Angle)

You can specify an angle override that locks the cursor for the next point entered.

To specify an angle override, enter a left angle bracket (<) followed by an angle whenever a command asks you to specify a point. The command prompt sequence below shows a 30-degree override entered during a LINE command.

Command: **line**
Specify first point: *Specify a start point for the line*
Specify next point or [Undo]: **<30**
Angle Override: **30**
Specify next point or [Undo]: *Specify a point*

The angle you specify will lock the cursor, overriding Grid Snap, Ortho mode, and PolarSnap. Coordinate entry and object snaps have precedence over an angle override.

Quick Reference

Commands

No entries

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Combine or Offset Points and Coordinates

To specify a new point location, you can combine coordinate values from several points or you can specify offsets from existing objects.

Combine Coordinate Values (Coordinate Filters)

You can use coordinate filters to extract one coordinate value at a time from locations on existing objects.

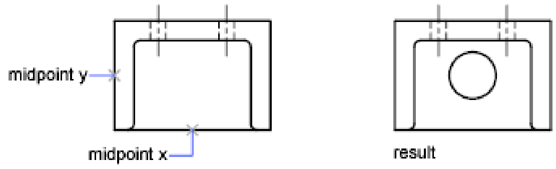
Coordinate filters specify a new coordinate location by using the *X* value from one location, the *Y* value of a second location, and, for 3D coordinates, the *Z* value of a third location. When used with object snaps, coordinate filters extract coordinate values from an existing object.

Coordinate filters are commonly used to locate the center of a rectangle and to locate the projection of a 3D point on the *XY* plane of the UCS.

To specify a filter at the command prompt, enter a period and one or more of the letters *X*, *Y*, and *Z*. The next entry is limited to a specific coordinate value.

Example: Use of Coordinate Filters in 2D

In the following illustration, the hole in the holding plate was centered in the rectangle by extracting the *X,Y* coordinates from the midpoints of the plate's horizontal and vertical line segments.



Here is the command prompt sequence:

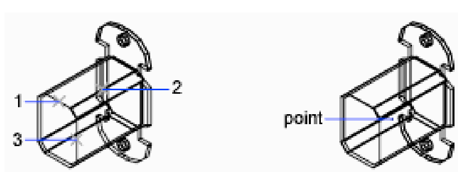
Command: **circle**
Specify center point for circle or [3P/2P/Ttr (tangent tangent radius)]: **.x**
of: **mid**
of: *Select the horizontal line on the lower edge of the holding plate*
of: (need YZ): **mid**
of: *Select the vertical line on the left side of the holding plate*
of: Diameter/<Radius> *Specify the radius of the hole*

Coordinate filters work only when the program prompts you for a point. If you try to use a coordinate filter at the command prompt, you see an error message.

Example: Use of Coordinate Filters in 3D

This example shows how to use coordinate filters to create a point object at the center (centroid) of a 3D object. Hidden lines have been removed for clarity. The *X* value of the new point is extracted from the first location specified, the *Y* value from the second location, and the *Z* value from the third. The three values are combined to form the coordinate values of the new point.

Command: **point**
Point: **.x**
of mid
of select object (*1*)
(need YZ): **.y**
of mid
of select object (*2)*
(need Z): **mid**
of select object (*3)*



To use coordinate filters to specify a point in 2D

1. At the prompt for a point, enter a coordinate filter (**.x** or **.y**).

   For example, enter **.x** to specify the *X* value first.

2. To extract the first coordinate value, specify a point.

   For example, if you entered **.x** in step 1, the *X* value is extracted from this point.

3. To extract the next coordinate value, specify a different point.

   The new point location combines the coordinate values extracted from the points you specified in steps 2 and 3.

NOTE Instead of specifying a point in steps 2 or 3, you can enter a numeric value.

To use coordinate filters to specify a point in 3D

1. At the prompt for a point, enter a coordinate filter (**.x**, **.y**, **.z**, **.xy**, **.xz**, or **.yz**).

   For example, enter **.x** to specify the *X* value first.

2. To extract the specified coordinate value(s), specify a point.

   For example, if you entered **.x** in step 1, the *X* value is extracted from this point.

3. At the prompt for the remaining coordinates, do one of the following:

   - Extract the remaining coordinate values by specifying a point.
   - Enter another coordinate filter and return to step 2.

   For example, if you entered **.x** in step 1, specify a second point to extract the *Y* and *Z* coordinates simultaneously, or enter **.y** or **.z** to specify *Y* and *Z* values separately.

   The new point location combines the coordinate values extracted from the points specified in steps 2 and 3.

NOTE Instead of specifying a point in steps 2 or 3, you can enter a numeric value.

Quick Reference

Commands

No entries

System Variables

No entries

Utilities

No entries

Command Modifiers

Coordinate Filters (Command Modifier)

Combines X, Y, and Z values from different points to specify a single point.

Track to Points on Objects (Object Snap Tracking)

You can draw objects at specific angles or in specific relationship to other objects along specified directions called alignment paths.

AutoTrack™ helps you draw objects at specific angles or in specific relationships to other objects. When you turn on AutoTrack, temporary *alignment* paths help you create objects at precise positions and angles. AutoTrack includes two tracking options: polar tracking and object snap tracking.

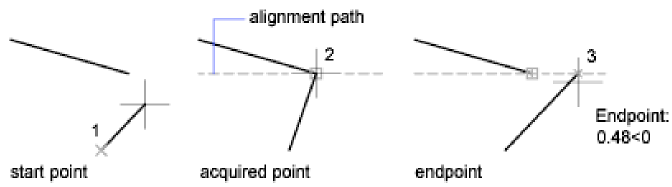
You can toggle AutoTrack on and off with the Polar and Otrack buttons on the status bar. Use temporary override keys to turn object snap tracking on and off or to turn off all snapping and tracking. See the keyboard illustration in Override Object Snap Settings on page 683.

Object snap tracking works in conjunction with object snaps. You must set an object snap before you can track from an object's snap point.

Object Snap Tracking

Use object snap tracking to track along alignment paths that are based on object snap points. Acquired points display a small plus sign (+), and you can acquire up to seven tracking points at a time. After you acquire a point, horizontal, vertical, or polar alignment paths relative to the point are displayed as you move the cursor over their drawing paths. For example, you can select a point along a path based on an object endpoint or midpoint or an intersection between objects.

In the following illustration, the Endpoint object snap is on. You start a line by clicking its start point (1), move the cursor over another line's endpoint (2) to acquire it, and then move the cursor along the horizontal alignment path to locate the endpoint you want for the line you are drawing (3).



Change Object Snap Tracking Settings

By default, object snap tracking is set to orthogonal. Alignment paths are displayed at 0, 90, 180, and 270 degrees from acquired object points. However, you can use polar tracking angles instead.

For object snap tracking, object points are automatically acquired. However, you can choose to acquire points only when you press SHIFT.

Change Alignment Path Display

You can change how AutoTrack displays alignment paths, and you can change how object points are acquired for object snap tracking. By default, alignment paths stretch to the end of the drawing window. You can change their display to abbreviated lengths, or no length.

Tips for Using Object Snap Tracking

As you use AutoTrack (polar tracking and object snap tracking), you will discover techniques that make specific design tasks easier. Here are a few you might try.

- Use Perpendicular, End, and Mid object snaps with object snap tracking to draw to points that are perpendicular to the end and midpoints of objects.
- Use the Tangent and End object snaps with object snap tracking to draw to points that are tangent to the endpoints of arcs.
- Use object snap tracking with temporary tracking points. At a point prompt, enter **tt**, then specify a temporary tracking point. A small + appears at the point. As you move your cursor, AutoTrack alignment paths are displayed

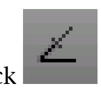
relative to the temporary point. To remove the point, move the cursor back over the +.

- After you acquire an object snap point, use direct distance to specify points at precise distances along alignment paths from the acquired object snap point. To specify a point prompt, select an object snap, move the cursor to display an alignment path, then enter a distance at the command prompt.

NOTE The direct distance entry method is not available while you are using the temporary override key for object snap tracking.

- Use the Automatic and SHIFT to Acquire options set on the Drafting tab of the Options dialog box to manage point acquisition. Point acquisition is set to Automatic by default. When working in close quarters, press SHIFT to temporarily avoid acquiring a point.

To turn on and turn off object snap tracking

- Press F11, or click  on the status bar.
  To turn object snap tracking on and off temporarily, hold down the F11 key while you work.

To change AutoTrack settings

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Drafting tab, under AutoTrack Settings, select or clear the following alignment path display options:
   - **Display Polar Tracking Vector.** Controls alignment path display for object snap tracking. When cleared, no polar tracking path is displayed.
   - **Display Full Screen Tracking Vector.** Controls alignment path display for object snap tracking. When cleared, an alignment path is displayed only from the object snap point to the cursor.
   - **Display AutoTrack Tooltip.** Controls the display of AutoTrack tooltips. Tooltips tell you the type of object snap (for object snap tracking), alignment angle, and distance from the previous point.

3 Under Alignment Point Acquisition, select a method for acquiring object points for object snap tracking:

- **Automatic.** Acquires object points automatically. If you select this option, you can press SHIFT to not acquire an object point
- **Shift to Acquire.** Acquires object points only when you press SHIFT while the cursor is over an object snap point.

Command entry: OPTIONS

Quick Reference

Commands

DSETTINGS

Sets grid and snap, polar and object snap tracking, object snap modes, Dynamic Input, and Quick Properties.

OPTIONS

Customizes the program settings.

System Variables

AUTOSNAP

Controls the display of the AutoSnap marker, tooltip, and magnet.

POLARMODE

Controls settings for polar and object snap tracking.

TRACKPATH

Controls the display of polar and object snap tracking alignment paths.

Utilities

No entries

Command Modifiers

No entries

Track to Offset Point Locations (Tracking)

You can use tracking to specify a point by offsetting vertically and horizontally from a series of temporary points.

You can use the tracking method whenever you are prompted for a point. Tracking uses the pointing device to specify a point by offsetting vertically and horizontally from a series of temporary points. When you start tracking and specify an initial reference point, the next reference point is constrained to a path that extends vertically or horizontally from that point. The direction of the offset is indicated by the rubber-band line. You change the direction of the offset by moving the cursor through the reference point. You can track as many points as you need. Typically, you use tracking in combination with object snaps or direct distance entry.

For example, you can use tracking to find the center point of a rectangle without using construction lines. Start tracking, and specify the midpoint of a horizontal line. Drag the cursor vertically and specify the midpoint of a vertical line (2). Press ENTER to accept the point (3) at the center of the rectangle.

To use tracking to specify a point

1. Start a command, such as the LINE command.
2. Hold down SHIFT and right-click in the drawing area. Click Tracking.
3. Specify a point.
4. Move the cursor directly up, down, left, or right until you see the rubber-band line.

   The direction of movement affects the tracking direction. Notice that if you move the cursor from left to right, you must then move it directly over the last point specified in order to move it up or down.
5. Specify a second point.
6. Press ENTER to end tracking.

The start point of the line snaps to the imaginary intersection of the vertical and horizontal paths extending from the points you specified. The position is determined by the direction in which you moved the cursor after specifying the first point.

Command entry: TRACKING (Command Modifier) (Command Modifier)

Quick Reference

Commands

No entries

System Variables

No entries

Utilities

No entries

Command Modifiers

TRACKING (Command Modifier)

Locates a point from a series of temporary points.

Specify Distances

When specifying a point, you can enter distances, offsets, and measured intervals.

Enter Direct Distances

You can specify a point by moving the cursor to indicate a direction and then entering the distance.

To specify a line length quickly, without entering coordinate values, you can specify a point by moving the cursor to indicate a direction and then entering the distance from the first point. You can enter calculated distances from the QuickCalc calculator. For more information, see Use the QuickCalc Calculator on page 773.

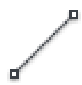
You can use direct distance entry to specify points for all commands requiring more than one point. When Ortho mode or polar tracking is on, this method is an efficient way to draw lines of specified length and direction, and to move or copy objects.

NOTE The direct distance entry method is not available while you are using the temporary override keys for Ortho mode, object snap tracking, or polar tracking.

See also:

- Use Polar Tracking and PolarSnap on page 695
- Lock an Angle for One Point (Angle) on page 700

To draw a line using direct distance entry

1. Click Home tab ➤ Draw panel ➤ Line. 
2. Specify the first point and then, move the pointing device until the rubber-band line extends at the same angle as the line you want to draw.
3. Enter a distance at the command prompt.

   The line is drawn at the length and angle you specified.
4. LINE

Quick Reference

Commands

LINE

Creates straight line segments.

System Variables

No entries

Utilities

No entries

Command Modifiers

Direct Distance Entry (Command Modifier)

Locates the next point at a specified distance in the direction of your cursor.

Offset from Temporary Reference Points

You can establish a temporary reference point as a base point for offsetting subsequent points.

The From command modifier establishes a temporary reference point as a base point for offsetting subsequent points. The From method does not constrain the cursor to orthogonal movement. The From method usually is used in combination with object snaps.

To offset a point from a temporary reference point

1. At a prompt for a point, enter **from**. Alternately, press SHIFT and right-click to display the object snap menu, and then choose From.
2. If you want to offset from a location on an existing object, specify an object snap method. Then select the object snap point.
3. Enter a relative coordinate (i.e. @1,1).

Quick Reference

Commands

No entries

System Variables

No entries

Utilities

No entries

Command Modifiers

FROM (Command Modifier)

Locates a point offset from a reference point within a command.

Specify Intervals on Objects

You can mark off equal distances along objects.

Overview of Specifying Intervals on Objects

Provides a high-level overview of two options for marking off equal distances along objects.

Sometimes you need to create points or insert symbols (blocks) at intervals on an object.

You can

- Specify the length of the segments (MEASURE)
- Specify the number of equal segments (DIVIDE)

You can measure or divide lines, arcs, splines, circles, ellipses, and polylines. With both methods, you can identify the intervals by inserting either a point or a block.

By specifying points, you can use the Node object snap to align other objects at intervals on the measured or divided object. By specifying blocks, you can create precise geometric constructions or insert custom markers. The blocks can rotate at each insertion point.

You cannot insert a block unless it has already been defined within the drawing. Variable attributes within the block are not included when you insert the block references.

The points or blocks you draw using MEASURE or DIVIDE are placed in a selection set. Therefore, if you want to edit them immediately, you can use the Previous option of *SELECT*.

See also:

- Create and Use Blocks (Symbols) on page 853

Quick Reference

Commands

BLOCK

Creates a block definition from selected objects.

DDPTYPE

Specifies the display style and size of point objects.

DIVIDE

Creates evenly spaced point objects or blocks along the length or perimeter of an object.

MEASURE

Creates point objects or blocks at measured intervals along the length or perimeter of an object.

WBLOCK

Writes objects or a block to a new drawing file.

System Variables

PDMODE

Controls how point objects are displayed.

PDSIZE

Sets the display size for point objects.

Utilities

No entries

Command Modifiers

No entries

Specify Measured Intervals on Objects

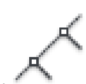
You can mark off equal lengths from one end of a selected object.

You can use MEASURE to mark an object at specified intervals. You can mark the intervals with either points or blocks. The last segment of a measured object may be shorter than the interval you specify.

The starting point for measurements or divisions varies with the object type. For lines or open polylines, the starting point is the endpoint closest to the selection point. For closed polylines, it is the polyline start point. For circles, it is at the angle from the center point that is equivalent to the current snap angle. For example, if the snap angle is 0, the circle starts at the three o'clock position and continues counterclockwise.

If the point marker is displayed as a single dot (the default setting), you may not be able to see the measured intervals. You can change the style of the point markers using several methods. To change the point style in a dialog box, you can use DDPTYPE. Alternately, click Format menu ➤ Point Style. The PDMODE system variable also controls the appearance of point markers. For example, you can change the value to make points appear as crosses. PDSIZE controls the size of point objects.

To insert points at measured intervals on an object

1 Click Home tab ➤ Draw panel ➤ Point drop-down ➤ Measure. 

2 Select a line, arc, spline, circle, ellipse, or polyline.

3 Enter an interval length, or specify points to indicate a length.

   Points are placed on the object at the specified intervals.

Command entry: MEASURE

To insert blocks at measured intervals on an object

1 If necessary, create the block you want to insert.

2 Click Home tab ➤ Draw panel ➤ Point drop-down ➤ Measure.

3 Select a line, arc, spline, circle, ellipse, or polyline.

4 Enter **b** (Block).

5 Enter the name of the block you want to insert.

6 Enter **y** to align the blocks with the measured object. Enter **n** to use a rotation angle of 0 degrees.

7 Enter an interval length, or specify points to indicate a length.

   Blocks are inserted on the object at the specified intervals.

Command entry: BLOCK, MEASURE

Quick Reference

Commands

BLOCK

Creates a block definition from selected objects.

DDPTYPE

Specifies the display style and size of point objects.

MEASURE

Creates point objects or blocks at measured intervals along the length or perimeter of an object.

System Variables

PDMODE

Controls how point objects are displayed.

PDSIZE

Sets the display size for point objects.

Utilities

No entries

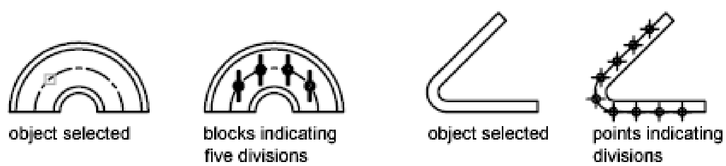
Command Modifiers

No entries

Divide an Object into Equal Segments

You can divide a selected object into a specified number of equal lengths.

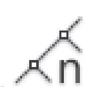
You can create points or insert blocks on an object at a specific number of equal intervals. This operation does not actually break an object into individual objects; it only identifies the location of the divisions so that you can use them as geometric reference points.



The starting point for measurements or divisions varies with the object type. For lines or open polylines, the starting point is the endpoint closest to the selection point. For closed polylines, it is the polyline start point. For circles, it is at the angle from the center point that is equivalent to the current snap angle. For example, if the snap angle is 0, the circle starts at the three o'clock position and continues counterclockwise.

If the point marker is displayed as a single dot (the default setting), you may not be able to see the segments. You can change the style of the point markers using several methods. To change the point style in a dialog box, you can use DDPTYPE. Alternately, click Format menu ➤ Point Style. The PDMODE system variable also controls the appearance of point markers. For example, you can change the value to make points appear as crosses. PDSIZE controls the size of point objects.

To insert points to mark equal segments

1 Click Home tab ➤ Draw panel ➤ Point drop-down ➤ Divide. 

2 Select a line, circle, ellipse, polyline, arc, or spline.

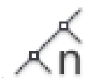
3 Enter the number of segments you want.

   A point is placed between each segment.

Command entry: DIVIDE

To insert blocks to mark equal segments on an object

1 If necessary, create the block you want to insert.

2 Click Home tab ➤ Draw panel ➤ Point drop-down ➤ Divide. 

3 Select a line, arc, circle, ellipse, polyline, or spline.

4 Enter **b** (Block).

5 Enter the name of the block you want to insert.

6 Enter **y** to align the blocks with the divided object. Enter **n** to use a rotation angle of 0 degrees.

7 Enter the number of segments you want.

Command entry: BLOCK, DIVIDE

Quick Reference

Commands

BLOCK

Creates a block definition from selected objects.

DDPTYPE

Specifies the display style and size of point objects.

DIVIDE

Creates evenly spaced point objects or blocks along the length or perimeter of an object.

WBLOCK

Writes objects or a block to a new drawing file.

System Variables

PDMODE

Controls how point objects are displayed.

PDSIZE

Sets the display size for point objects.

Utilities

No entries

Command Modifiers

No entries

Design with Parametric Constraints

With parametric drawing, you can add constraints to geometry to ensure that the design conforms to specified requirements.

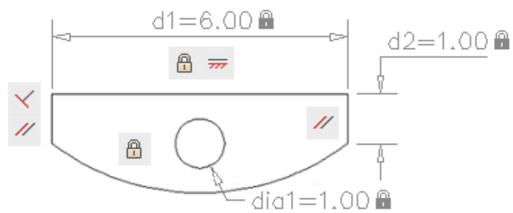
Overview of Constraints

Parametric drawing is a technology that is used for designing with constraints. *Constraints* are associations and restrictions applied to 2D geometry.

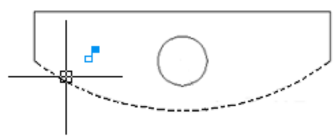
There are two general types of constraints:

- *Geometric constraints* control the relationships of objects with respect to each other
- *Dimensional constraints* control the distance, length, angle, and radius values of objects

The following illustration displays geometric and dimensional constraints using the default format and visibility.



A blue cursor icon always displays when you move the cursor over an object that has constraints applied to it.



In the design phase of a project, constraints provide a way to enforce requirements when experimenting with different designs or when making changes. Changes made to objects can adjust other objects automatically, and restrict changes to distance and angle values.

With constraints, you can

- Maintain design specifications and requirements by constraining the geometry within a drawing
- Apply multiple geometric constraints to objects instantly
- Include formulas and equations within dimensional constraints
- Make design changes quickly by changing the value of a variable

---

BEST PRACTICE It is recommended that you first apply geometric constraints to determine the *shape* of a design, and then apply dimensional constraints to determine the *size* of objects in a design.

Design Using Constraints

When you are creating or changing a design, a drawing will be in one of three states:

- *Unconstrained.* No constraints are applied to any geometry.

- *Underconstrained*. Some constraints are applied to the geometry.
- *Fully constrained*. All relevant geometric and dimensional constraints are applied to the geometry. A fully constrained set of objects also needs to include at least one Fix constraint to lock the location of the geometry.

Thus, there are two general methods for designing with constraints:

- You can work in an underconstrained drawing and make changes as you go, using a combination of editing commands, grips, and adding or changing constraints.
- You can create and fully constrain a drawing first, and then control the design exclusively by relaxing and replacing geometric constraints, and changing the values in dimensional constraints.

The method that you choose depends on your design practices and the requirements of your discipline.

NOTE The program prevents you from applying any constraints that result in an overconstrained condition.

Use Constraints with Blocks and Xrefs

You can apply constraints between

- An object in the drawing and an object within a block reference
- An object within a block reference and an object within a *different* block reference (not between objects within the same block reference)
- The insertion point of an xref and an object or a block, but not to any objects within xrefs

When you apply constraints to block references, the objects contained within the block are automatically available for selection. You do not need to press Ctrl for subobject selection. Adding constraints to a block reference can cause it to move or rotate as a result.

NOTE Applying constraints to dynamic blocks suppresses the display of their dynamic grips. You can still change the values in a dynamic block using the Properties palette, but to redisplay the dynamic grips, the constraints must first be removed from the dynamic block.

Constraints can be used in block definitions, resulting in dynamic blocks. You can control the size and shape of dynamic blocks directly from within the drawing. For more information, see Add Constraints to Dynamic Blocks on page 898.

Remove or Relax Constraints

There are two ways to cancel the effects of constraints when you need to make design changes:

- Delete the constraints individually and later apply new constraints. While the cursor hovers over a geometric constraint icon, you can use the Delete key or the shortcut menu to delete the constraint.
- Relax the constraints temporarily on selected objects to make the changes. With a grip selected or when you specify options during an editing command, tap the Ctrl key to alternate between relaxing constraints and maintaining constraints.

Relaxed constraints are not maintained during editing. Constraints are restored automatically if possible when the editing process is complete. Constraints that are no longer valid are removed.

NOTE The DELCONSTRAINT command deletes all geometric and dimensional constraints from an object.

Quick Reference

Commands

AUTOCONSTRAIN

Applies geometric constraints to a selection set of objects based on orientation of the objects relative to one another.

CONSTRAINTBAR

A toolbar-like UI element that displays the available geometric constraints on an object.

CONSTRAINTSETTINGS

Controls the display of geometric constraints on constraint bars.

DELCONSTRAINT

Removes all geometric and dimensional constraints from a selection set of objects.

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects.

GEOMCONSTRAINT

A toolbar-like UI element that displays the available geometric constraints on an object.

LIST

Displays property data for selected objects.

PARAMETERS

Controls the associative parameters used in the drawing.

-PARAMETERS

Controls the associative parameters used in the drawing.

PARAMETERSCLOSE

Closes the Parameters Manager palette.

TEXTEDIT

Edits a dimensional constraint, dimension, or text object.

System Variables

CCONSTRAINTFORM

Controls whether annotational or dynamic constraints are applied to objects.

CONSTRAINTBARDISPLAY

Displays constraint bars for objects after you manually apply a constraint or autoconstrain them.

CONSTRAINTBARMODE

Controls the display of geometrical constraints on constraint bars.

CONSTRAINTNAMEFORMAT

Controls the text format for dimensional constraints.

CONSTRAINTRELAX

Indicates whether constraints are enforced or relaxed when editing an object.

CONSTRAINTSOLVEMODE

Controls constraint behavior when applying or editing constraints.

DIMCONSTRAINTICON

Displays the lock icon next to the text for dimensional constraints.

DYNCONSTRAINTDISPLAY

Displays or hides dynamic constraints.

DYNCONSTRAINTMODE

Displays hidden dimensional constraints when constrained objects are selected.

PARAMETERCOPYMODE

Controls how constraints and referenced variables are copied when replicating constrained geometry.

PARAMETERSSTATUS

Indicates whether the Parameters Manager palette is displayed or hidden.

Constrain Objects Geometrically

Geometric constraints determine the relationships between 2D geometric objects or points on objects relative to each other.

Overview of Geometric Constraints

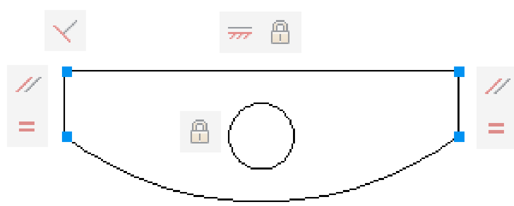
You can specify geometric constraints between 2D objects or points on objects. When you later edit the constrained geometry, the constraints are maintained.

Thus, using geometric constraints, you have a method of including design requirements in your drawing.

For example, in the illustration below, the following constraints are applied to the geometry.

- Every endpoint is constrained to remain coincident with the endpoint of every adjacent object—these constraints are displayed as small blue squares

- The vertical lines are constrained to remain parallel with each other and to remain equal to each other in length
- The left vertical line is constrained to remain perpendicular to the horizontal line
- The horizontal line is constrained to remain horizontal
- The location of the circle and the horizontal line are constrained to remain fixed in space—these constraints are displayed as lock icons



The geometry is not *fully constrained*, however. Using grips, you can still change the radius of the arc, the diameter of the circle, the length of the horizontal line, and the length of the vertical lines. To specify these distances, you need to apply dimensional constraints.

NOTE Constraints can be added to segments within a polyline as if they were separate objects.

See also:

- Overview of Dimensional Constraints on page 741

Quick Reference

Commands

AUTOCONSTRAIN

Applies geometric constraints to a selection set of objects based on orientation of the objects relative to one another.

CONSTRAINTBAR

A toolbar-like UI element that displays the available geometric constraints on an object.

CONSTRAINTSETTINGS

Controls the display of geometric constraints on constraint bars.

DELCONSTRAINT

Removes all geometric and dimensional constraints from a selection set of objects.

GEOMCONSTRAINT

A toolbar-like UI element that displays the available geometric constraints on an object.

LIST

Displays property data for selected objects.

System Variables

CONSTRAINTBARMODE

Controls the display of geometrical constraints on constraint bars.

CONSTRAINTBARDISPLAY

Displays constraint bars for objects after you manually apply a constraint or autoconstrain them.

CONSTRAINTNAMEFORMAT

Controls the text format for dimensional constraints.

CONSTRAINTRELAX

Indicates whether constraints are enforced or relaxed when editing an object.

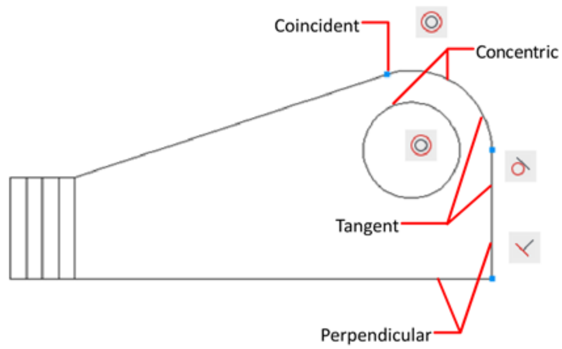
CONSTRAINTSOLVEMODE

Controls constraint behavior when applying or editing constraints.

Apply or Remove Geometric Constraints

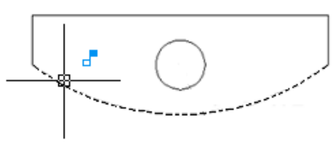
Geometric constraints associate geometric objects together, or specify a fixed location or angle.

For example, you can specify that a line should always be perpendicular to another one, that an arc and a circle should always remain concentric, or that a line should always be tangent to an arc.



When you apply a constraint, two things occur:

- The object that you select adjusts automatically to conform to the specified constraint
- By default, a gray constraint icon displays near the constrained object as shown in the previous illustration, and a small, blue glyph displays with your cursor when you move it over a constrained object



Once applied, constraints permit only those changes to the geometry that do not violate the constraints. This provides a method for exploring design options or making design changes while maintaining the requirements and specifications of the design.

NOTE The order in which you select two objects when you apply a constraint is important in some cases. Normally, the second object you select adjusts to the first object. For example, when you apply a Perpendicular constraint, the second object you select will adjust to become perpendicular to the first.

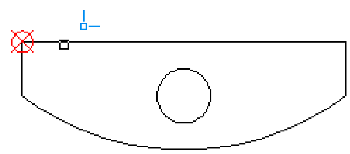
You can apply geometric constraints to 2D geometric objects only. Objects cannot be constrained between model space and paper space.

Specify Constraint Points

With some constraints, you specify *constraint points* on objects instead of selecting the objects. This behavior is similar to that of object snaps, but the locations are limited to endpoints, midpoints, center points, and insertion points.

For example, the Coincident constraint can restrict the location of the endpoint of one line to the endpoint of another line.

The following glyph is displayed on the object as you roll over the object.



You use this glyph to confirm whether you are specifying the intended point to constrain.

Use Fix Constraints

A Fix constraint associates a constraint point on an object, or the object itself with a fixed location with respect to the World Coordinate System.

It is often advisable to specify a Fix constraint at an important geometric feature. This locks the location of that point or object, and prevents geometry from relocating when you make changes to the design.

When you fix an *object*, the angle of a line, or the center of an arc or circle is also fixed.

Apply Multiple Geometric Constraints

You can apply multiple geometric constraints to objects either manually or automatically.

When you want to apply all essential geometric constraints to a design automatically, you can use AUTOCONSTRAIN with the objects that you select in your drawing. This helps constrain the geometric shape of the design—depending on your design, there might be cases where you need to apply additional geometric constraints.

AUTOCONSTRAIN also provides settings in which you can specify the following options:

- What geometric constraints to apply

- What order to apply geometric constraints
- What tolerances are used to determine whether objects are horizontal, vertical, or touching

---

NOTE Equal or Fix constraints are not applied with AUTOCONSTRAIN. You must apply these constraints individually.

To fully constrain the size and proportions of a design, you will later need to apply dimensional constraints.

Remove Geometric Constraints

A geometric constraint cannot be modified, but you can delete it and apply a different one. Several constraint options, including Delete, are available from the shortcut menu that is displayed when you right-click a constraint icon in the drawing.

You can delete all constraints from a selection set in a single operation with DELCONSTRAINT.

To apply a coincident constraint

1. Click Parametric tab ➤ Geometric panel ➤ Coincident.
2. Select the first and second points on two different objects.

   The second point is made coincident to the first.

Toolbar: Parametric, Geometric Constraints
Command entry: GEOMCONSTRAINT

To apply a collinear constraint

1. Click Parametric tab ➤ Geometric panel ➤ Collinear.
2. Select the first and second object. You can select either a line object or a polyline subobject.

   The second object is made collinear to the first.

Toolbar: Parametric, Geometric Constraints
Command entry: GEOMCONSTRAINT

To apply a concentric constraint

1 Click Parametric tab ➤ Geometric panel ➤ Concentric.

2 Select the first and second arc or circle object.

The second arc or circle moves to have the same center point as the first object.

Toolbar: Parametric, Geometric Constraints
Command entry: GEOMCONSTRAINT

To apply a fix constraint

1 Click Parametric tab ➤ Geometric panel ➤ Fix.

2 Select a point on an object.

Applying the Fix constraint to a point on the object locks the node in place. You can still move the object.

Toolbar: Parametric, Geometric Constraints
Command entry: GEOMCONSTRAINT

To apply a parallel constraint

1 Click Parametric tab ➤ Geometric panel ➤ Parallel.

2 Select the two objects to be made parallel. You can select either a line object or a polyline subobject.

The second object is made parallel with the first object.

Toolbar: Parametric, Geometric Constraints
Command entry: GEOMCONSTRAINT

To apply a perpendicular constraint

1 Click Parametric tab ➤ Geometric panel ➤ Perpendicular.

2 Select the two objects to be made perpendicular. You can select either a line object or a polyline subobject.

   The second object is made perpendicular to the first object.

Toolbar: Parametric, Geometric Constraints
Command entry: GEOMCONSTRAINT

To apply a horizontal constraint

1 Click Parametric tab ➤ Geometric panel ➤ Horizontal.

2 Select the line object or polyline subobject to be made horizontal.

Toolbar: Parametric, Geometric Constraints
Command entry: GEOMCONSTRAINT

To apply a vertical constraint

1 Click Parametric tab ➤ Geometric panel ➤ Vertical.

2 Select the line object or polyline subobject to be made vertical.

Toolbar: Parametric, Geometric Constraints
Command entry: GEOMCONSTRAINT

To apply a tangent constraint

1 Click Parametric tab ➤ Geometric panel ➤ Tangent.

2 Select the two objects to be made tangential.

The second object maintains a point of tangency with the first object.

Toolbar: Parametric, Geometric Constraints
Command entry: GEOMCONSTRAINT

To apply a smooth constraint

1 Click Parametric tab ➤ Geometric panel ➤ Smooth.

2 Select the first spline curve.

3 Select the second spline, line, polyline (subobject), or arc object.

The two objects are updated to be contiguous with one another.

Toolbar: Parametric, Geometric Constraints
Command entry: GEOMCONSTRAINT

To apply a symmetric constraint

1 Click Parametric tab ➤ Geometric panel ➤ Symmetric.

2 Select the first and second object.

3 Select the symmetry line.

The selected objects become symmetrically constrained about the selected line.

Toolbar: Parametric, Geometric Constraints
Command entry: GEOMCONSTRAINT

To apply an equal constraint

1 Click Parametric tab ➤ Geometric panel ➤ Equal.

2 Select the first and second object.

The second object is made equal to the first object.

Toolbar: Parametric, Geometric Constraints
Command entry: GEOMCONSTRAINT

To apply multiple geometric constraints to an object

1 Click Parametric tab ➤ Geometric panel ➤ AutoConstrain.

2 Select the objects that you want to constrain.

3 Press Enter when you select the objects to be automatically constrained.

The Command prompt displays the number of constraints applied.

Toolbar: Parametric, Geometric Constraints
Command entry: AUTOCONSTRAIN

To set the order for applying multiple geometric constraints to an object

1 Click Parametric tab ➤ Geometric panel ➤ AutoConstrain.

2 At the Command prompt, enter **s** (Settings).

3 In the Constraint Settings dialog box, on the AutoConstrain tab, select a Constraint Type.

4 Click Move Up or Move Down. This changes the priority for a constraint when you use the AUTOCONSTRAIN command on an object.

5 Click OK.

Toolbar: Parametric, Geometric Constraints 
Command entry: AUTOCONSTRAIN

Quick Reference

Commands

GEOMCONSTRAINT

A toolbar-like UI element that displays the available geometric constraints on an object.

CONSTRAINTBAR

A toolbar-like UI element that displays the available geometric constraints on an object.

CONSTRAINTSETTINGS

Controls the display of geometric constraints on constraint bars.

System Variables

CONSTRAINTBARMODE

Controls the display of geometrical constraints on constraint bars.

CONSTRAINTBARDISPLAY

Displays constraint bars for objects after you manually apply a constraint or autoconstrain them.

CONSTRAINTNAMEFORMAT

Controls the text format for dimensional constraints.

CONSTRAINTRELAX

Indicates whether constraints are enforced or relaxed when editing an object.

CONSTRAINTSOLVEMODE

Controls constraint behavior when applying or editing constraints.

Display and Verify Geometric Constraints

You can determine visually what objects are associated with any geometric constraint, or what constraints are associated with any object.

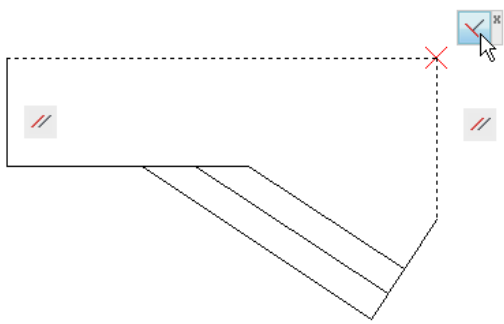
Constraint icons provide information about how objects are constrained. A *constraint bar* displays one or more icons that represent the geometric constraints applied to an object.

You can drag constraint bars when you need to move them out of the way, and you can also control whether they are displayed or hidden.

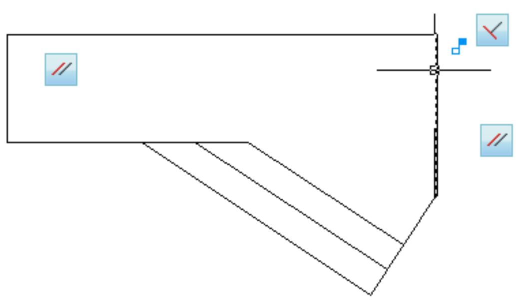
Verify the Geometric Constraints on Objects

You can confirm the association of geometric constraints with objects in two ways.

- When you roll over a *constraint icon* on a constraint bar, the objects associated with that geometric constraint are highlighted.



- When you roll over an *object* that has geometric constraints applied to it, all constraint bars that are associated with the object are highlighted.



These highlighting features simplify working with constraints especially when you have many constraints applied throughout a drawing.

Control the Display of Constraint Bars

Geometric constraints and constraint bars can be displayed or hidden, either individually or globally. You can do any of the following:

- Display or hide all geometric constraints
- Display or hide specified types of geometric constraints
- Display or hide all geometric constraints associated with a selected object

Use the Constraint Settings dialog box to control the types of geometric constraints that are displayed or hidden on constraint bars.

Hiding geometric constraints is useful when you analyze a design and want to filter the display of geometric constraints. For example, you can choose to display the icons for Parallel constraints only. Next, you might choose to display the icons for Perpendicular constraints only.

When you are not working with geometric constraints, it is recommended that you hide them globally.

NOTE To reduce clutter, Coincident constraints display by default as small, light-blue squares. You can use an option in the Constraint Settings dialog box to turn them off if necessary.

To display a geometric constraint

1. Click Parametric tab ➤ Geometric panel ➤ Show. 
2. Select the constrained objects.
3. Press Enter.

Toolbar: parametric 
Command entry: CONSTRAINTBAR

To display all geometric constraints

➤ Click Parametric tab ➤ Geometric panel ➤ Show All.

Toolbar: Parametric
Command entry: CONSTRAINTBAR

To hide all geometric constraints

➤ Click Parametric tab ➤ Geometric panel ➤ Hide All.

Toolbar: Parametric
Command entry: CONSTRAINTBAR

To change the constraint bar settings using the constraint bar shortcut menu

1 Select a constrained object.

2 Ensure that the constraint bar is visible for the selected object.

3 Right-click the constraint bar, and click Constraint Bar Settings.

4 In the Constraint Settings dialog box, on the Geometric tab, select or clear the appropriate check boxes.

5 Use the slider, or enter a value, to set the transparency level of constraint bars in the drawing. The default value is 50.

6 Click OK.

Toolbar: Parametric
Command entry: CONSTRAINTBAR

Quick Reference

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Controls the text format for dimensional constraints.

CONSTRAINTRELAX

Indicates whether constraints are enforced or relaxed when editing an object.

CONSTRAINTSOLVEMODE

Controls constraint behavior when applying or editing constraints.

Modify Objects with Geometric Constraints Applied

You can edit constrained geometric objects with grips, editing commands, or by relaxing or applying geometric constraints.

By definition, geometric constraints that are applied to geometric objects limit the editing actions that you perform on the objects.

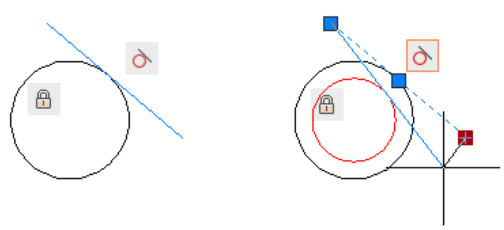
Modify Constrained Objects with Grips

You can modify constrained geometry using grip editing modes. The geometry will maintain all applied constraints.

For example, if a line object is constrained to remain tangent to a circle, you can rotate the line and change its length and endpoints, but the line *or its extension* will remain tangent to the circle.

If the circle was an arc instead, the line *or its extension* would remain tangent to the arc *or its extension*.

The results of modifying underconstrained objects are based on what constraints have already been applied and the object types involved. For example, if the Radius constraint had not been applied, the radius of the circle would have been modified instead of the tangent point of the line.



The CONSTRAINTSOLVEMODE system variable determines the way an object behaves when constraints are applied or when grips are used to edit it.

BEST PRACTICE You can limit unexpected changes by applying additional geometric or dimensional constraints. Common choices include Coincident and Fix constraints.

Modify Constrained Objects with Editing Commands

You can use editing commands such as MOVE, COPY, ROTATE, and SCALE to modify constrained geometry. The results maintain the constraints applied to the objects.

NOTE The TRIM, EXTEND, BREAK, and JOIN commands in some circumstances can remove constraints.

By default, if an editing command results in copying the constrained objects, the constraints applied to the original objects will also be duplicated. This behavior is controlled by the PARAMETERCOPYMODE system variable. Using the copying technique, you can save work by taking advantage of multiple instances of objects, bilateral symmetry, or radial symmetry.

For information about temporarily relaxing constraints, see Overview of Constraints on page 718.

To grip-edit constrained geometry

1 Select the constrained object.

2 Click the grips and drag it to edit the geometry.

Toolbar: Parametric, Geometric Constraints
Command entry: GEOMCONSTRAINT

To turn a constraint off

1 Click the constrained object to select it.

2 Move your mouse over a grip.
The grips are displayed in red to show that the object is selected.

3 Click the grip.

4 Press and release the Ctrl key.

5 Move the object. The object moves freely as it is no longer constrained.
Constraint bars will no longer be displayed (if enabled) for the object, as the constraints are turned off.

Toolbar: Parametric, Geometric Constraints

Command entry: GEOMCONSTRAINT

To delete a geometric constraint

1. Select a constrained object.
2. Ensure that the constraint bar is visible for the selected object.
3. Right-click the constraint bar. Click Delete.

   Constraint bars for the deleted constraint will no longer display for the object.

Toolbar: Parametric
Command entry: DELCONSTRAINT

To delete all geometric constraints from an object

1. Select a constrained object.
2. Click Parametric tab ➤ Manage panel ➤ Delete Constraint.

   NOTE All geometric and dimensional constraints are removed from the object.

   ---
3. Press Enter.

Toolbar: Parametric
Command entry: DELCONSTRAINT

Quick Reference

Commands

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A toolbar-like UI element that displays the available geometric constraints on an object.

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Controls the text format for dimensional constraints.

CONSTRAINTRELAX

Indicates whether constraints are enforced or relaxed when editing an object.

CONSTRAINTSOLVEMODE

Controls constraint behavior when applying or editing constraints.

Constrain Distances and Angles between Objects

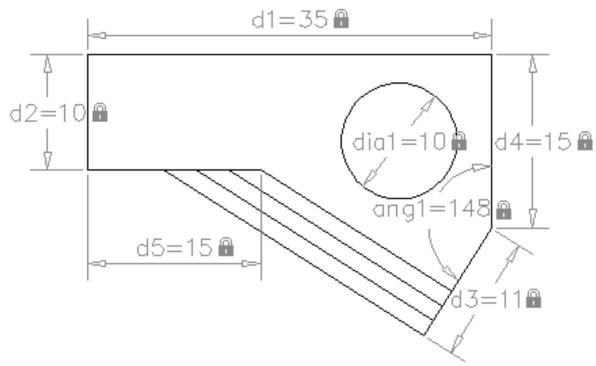
You can control distances or angles between 2D geometric objects or points on objects applying dimensional constraints and specifying values. You can also constrain geometry with variables and equations.

Overview of Dimensional Constraints

Dimensional constraints control the size and proportions of a design. They can constrain the following:

- Distances between objects, or between points on objects
- Angles between objects, or between points on objects
- Sizes of arcs and circles

For example, the following illustration includes linear, aligned, angular, and diameter constraints.



If you change the value of a dimensional constraint, *all* the constraints on the object are evaluated, and the objects that are affected are updated automatically.

Also, constraints can be added directly to segments within a polyline as if they were separate objects.

NOTE The number of decimal places displayed in dimensional constraints is controlled by the LUPREC and AUPREC system variables.

Compare Dimensional Constraints with Dimension Objects

Dimensional *constraints* are different from dimension *objects* in the following ways:

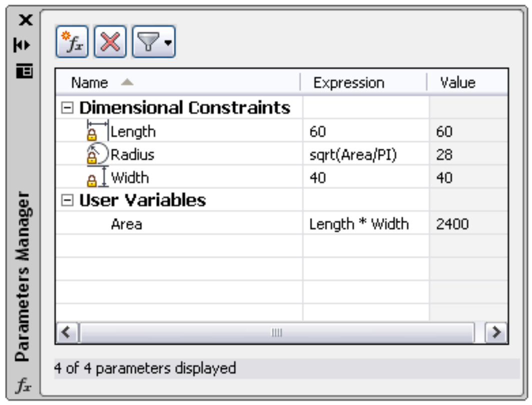
- Dimensional constraints are used in the design phase of a drawing, but dimensions are typically created in the documentation phase
- Dimensional constraints *drive* the size or angle of objects, but dimensions *are driven* by objects
- By default, dimensional constraints are not objects, display with only a single dimension style, maintain the same size during zoom operations, and are not plotted

If you need to plot dimensional constraints or use dimension styles, you can change the form of a dimensional constraint from dynamic to annotational. See Apply Dimensional Constraints on page 744 for more detail.

Define Variables and Equations

With the *Parameters Manager*, you can define custom *user variables* that you can reference from within dimensional constraints and other user variables.

The expressions that you define can include a variety of predefined functions and constants.



For more information about using variables and equations with constraints, see Constrain a Design with Formulas and Equations on page 758

See also:

- Overview of Geometric Constraints on page 723

Quick Reference

Commands

DELCONSTRAINT

Removes all geometric and dimensional constraints from a selection set of objects.

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects.

LIST

Displays property data for selected objects.

PARAMETERS

Controls the associative parameters used in the drawing.

-PARAMETERS

Controls the associative parameters used in the drawing.

PARAMETERSCLOSE

Closes the Parameters Manager palette.

TEXTEDIT

Edits a dimensional constraint, dimension, or text object.

System Variables

CCONSTRAINTFORM

Controls whether annotational or dynamic constraints are applied to objects.

CONSTRAINTNAMEFORMAT

Controls the text format for dimensional constraints.

CONSTRAINTRELAX

Indicates whether constraints are enforced or relaxed when editing an object.

CONSTRAINTSOLVEMODE

Controls constraint behavior when applying or editing constraints.

DIMCONSTRAINTICON

Displays the lock icon next to the text for dimensional constraints.

DYNCONSTRAINTDISPLAY

Displays or hides dynamic constraints.

DYNCONSTRAINTMODE

Displays hidden dimensional constraints when constrained objects are selected.

PARAMETERCOPYMODE

Controls how constraints and referenced variables are copied when replicating constrained geometry.

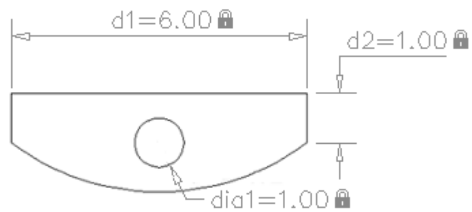
PARAMETERSSTATUS

Indicates whether the Parameters Manager palette is displayed or hidden.

Apply Dimensional Constraints

Dimensional constraints maintain specified distances and angles between geometric objects or points on objects.

For example, you can specify that the length of a line should always remain at 6.00 units, that the vertical distance between two points be maintained at 1.00 unit, and that a circle should always remain at 1.00 unit in diameter.



When you apply a dimensional constraint to an object, a *constraint variable* is automatically created for maintaining the constraint value. By default, these are assigned names such as *d1* or *dia1*, but you can rename them in the Parameters Manager.

Dimensional constraints can be created in one of the following forms:

- Dynamic constraints
- Annotational constraints

The forms have different purposes. In addition, any dynamic or annotational constraint can be converted to a *reference constraint*.

Dynamic Constraints

By default, dimensional constraints are *dynamic*. They are ideal for normal parametric drawing and design tasks.

Dynamic constraints have the following characteristics:

- Maintain the same size when zooming in or out
- Can easily be turned on or off globally in the drawing
- Display using a fixed, predefined dimension style
- Position the textual information automatically, and provide triangle grips with which you can change the value of a dimensional constraint
- Do not display when the drawing is plotted

If you need to control the dimension style of dynamic constraints, or if you need to plot dimensional constraints, use the Properties palette to change dynamic constraints to annotational constraints.

Annotational Constraints

Annotational constraints are useful when you want dimensional constraints to have the following characteristics:

- Change their size when zooming in or out
- Display individually with layers
- Display using the *current* dimension style
- Provide grip capabilities that are similar to those on dimensions
- Display when the drawing is plotted

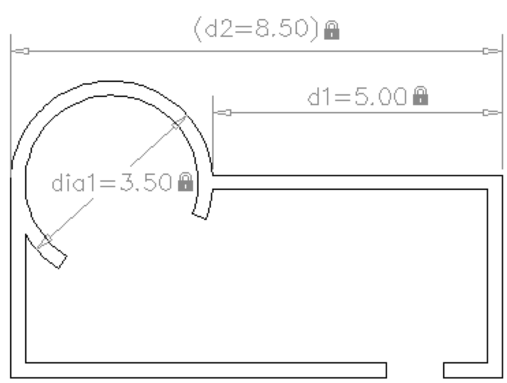
NOTE To display the text used in annotational constraints in the same format as used in dimensions, set the CONSTRAINTNAMEFORMAT system variable to 1.

After plotting, you can use the Properties palette to convert annotational constraints back to dynamic constraints.

Reference Constraints

A reference constraint is a *driven* dimensional constraint, either dynamic or annotational. This means that it does not control the associated geometry, but rather reports a measurement similar to a dimension object.

You use reference constraints as a convenient way to display measurements that you would otherwise have to calculate. For example, the width in the illustration is constrained by the diameter constraint, *dia1*, and the linear constraint, *d1*. The reference constraint, *d2*, displays the total width but does not constrain it. The textual information in reference constraints is always displayed within parentheses.



You can set the Reference property in the Properties palette to convert a dynamic or annotational constraint to a reference constraint.

NOTE You cannot change a reference constraint back to a dimensional constraint if doing so would overconstrain the geometry.

To create a linear dimensional constraint

1. Click Parametric tab ➤ Dimensional panel ➤ Linear.
2. Select the two constraint points on the object.
3. Specify the dimension line location.
4. Enter a value or specify an expression (name=value).
5. Press Enter.
   By default, a dynamic constraint is created.

Toolbar: Parametric, Dimensional Constraints
Command entry: DIMCONSTRAINT

To create a horizontal dimensional constraint

1. Click Parametric tab ➤ Dimensional panel ➤ Horizontal.
2. Select the two constraint points on the object.
3. Specify the dimension line location.
4. Enter a value or specify an expression (name=value).
5. Press Enter.
   By default, a dynamic constraint is created.

Toolbar: Parametric, Dimensional Constraints
Command entry: DIMCONSTRAINT

To create a vertical dimensional constraint

1. Click Parametric tab ➤ Dimensional panel ➤ Vertical.

2. Select the two constraint points on the object.
3. Specify the dimension line location.
4. Enter a value or specify an expression (name=value).
5. Press Enter.
   By default, a dynamic constraint is created.

Toolbar: Parametric, Dimensional Constraints
Command entry: DIMCONSTRAINT

To create an aligned dimensional constraint

1. Click Parametric tab ➤ Dimensional panel ➤ Aligned.
2. Select the two constraint points on the object.
3. Specify the dimension line location.
4. Enter a value or specify an expression (name=value).
5. Press Enter.
   By default, a dynamic constraint is created.

Toolbar: Parametric, Dimensional Constraints
Command entry: DIMCONSTRAINT

To create an angular dimensional constraint

1. Click Parametric tab ➤ Dimensional panel ➤ Angular.
2. Select two lines or an arc object.
3. Specify the dimension line location.
4. Enter a value or specify an expression (name=value).
5. Press Enter.
   By default, a dynamic constraint is created.

Toolbar: Parametric, Dimensional Constraints

Command entry: DIMCONSTRAINT

To create a radial dimensional constraint

1. Click Parametric tab ➤ Dimensional panel ➤ Radial.
2. Select an arc or a circle object.
3. Specify the dimension line location.
4. Enter a value or specify an expression (name=value).
5. Press Enter.
   By default, a dynamic constraint is created.

Toolbar: Parametric, Dimensional Constraints
Command entry: DIMCONSTRAINT

To create a diameter dimensional constraint

1. Click Parametric tab ➤ Dimensional panel ➤ Diameter.
2. Select an arc or a circle object.
3. Specify the dimension line location.
4. Enter a value or specify an expression (name=value).
5. Press Enter.
   By default, a dynamic constraint is created.

Toolbar: Parametric, Dimensional Constraints
Command entry: DIMCONSTRAINT

To convert a dynamic constraint to an annotational constraint

1. Select the dynamic constraint.
2. At the Command prompt, enter **properties**.
3. Click the down arrow at the right of the Constraint Form property and select Annotational.

The Properties palette is populated with additional properties as the constraint is now an annotational constraint.

Toolbar: Parametric, Dimensional Constraints
Command entry: DIMCONSTRAINT

To convert a dynamic or annotative constraint to a reference constraint

1. Select the dynamic or annotational constraint.
2. At the Command prompt, enter **properties**.
3. Click the down arrow at the right of the Reference property and select Yes.

   The Expression property is shaded to denote that it is not editable.

Toolbar: Parametric, Dimensional Constraints
Command entry: DIMCONSTRAINT

To change the dimension name format

1. Select an annotational constraint, right-click in the drawing area, and click Dimension Name Format.
2. Select Value, Name, or Name and Expression.

   The Expression reflects the selected dimension name format.

Toolbar: Parametric, Dimensional Constraints
Command entry: DIMCONSTRAINT

Quick Reference

Commands

DELCONSTRAINT

Removes all geometric and dimensional constraints from a selection set of objects.

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects.

LIST

Displays property data for selected objects.

PARAMETERS

Controls the associative parameters used in the drawing.

-PARAMETERS

Controls the associative parameters used in the drawing.

PARAMETERSCLOSE

Closes the Parameters Manager palette.

TEXTEDIT

Edits a dimensional constraint, dimension, or text object.

System Variables

CONSTRAINTNAMEFORMAT

Controls the text format for dimensional constraints.

CONSTRAINTRELAX

Indicates whether constraints are enforced or relaxed when editing an object.

CONSTRAINTSOLVEMODE

Controls constraint behavior when applying or editing constraints.

DIMCONSTRAINTICON

Displays the lock icon next to the text for dimensional constraints.

DYNCONSTRAINTDISPLAY

Displays or hides dynamic constraints.

DYNCONSTRAINTMODE

Displays hidden dimensional constraints when constrained objects are selected.

PARAMETERCOPYMODE

Controls how constraints and referenced variables are copied when replicating constrained geometry.

PARAMETERSSTATUS

Indicates whether the Parameters Manager palette is displayed or hidden.

Control the Display of Dimensional Constraints

You can display or hide dynamic and annotational constraints within a drawing.

Display or Hide Dynamic Constraints

You can hide all dynamic constraints globally within a drawing to reduce clutter when you want to work with geometric constraints only, or when you need to continue other work in the drawing. You can turn on their display when needed from the ribbon or with the DYNCONSTRAINTDISPLAY system variable.

By default, if you select an object associated with a hidden dynamic constraint, all dynamic constraints associated with that object will display.

Display or Hide Annotational Constraints

You control the display of annotational constraints as you would with dimension objects—you assign them to a layer and turn the layer on or off as needed. You can also specify object properties for annotational constraints such as dimension style, color, and lineweight.

To display dynamic dimensional constraints

➤ Click Parametric tab ➤ Dimensional panel ➤ Show Dynamic Constraints.

To hide dynamic dimensional constraints

1 Ensure that dynamic constraints are displayed.

2 Click Parametric tab ➤ Dimensional panel ➤ Show Dynamic Constraints

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DYNCONSTRAINTMODE

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PARAMETERCOPYMODE

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PARAMETERSSTATUS

Indicates whether the Parameters Manager palette is displayed or hidden.

Modify Objects with Dimensional Constraints Applied

You can control lengths, distances, and angles of objects by changing constraint values, by manipulating dimensional constraints using grips, or by changing user variables or expressions associated with dimensional constraints.

Edit Dimensional Constraint Names, Values, and Expressions

You can easily edit the names, values, and expressions that are associated with dimensional constraints. There are four access points that you can use:

- Double-click the dimensional constraint, select the dimensional constraint and use the shortcut menu, or the TEXTEDIT command.
- Open the Properties palette and select the dimensional constraint
- Open the Parameters Manager and select the dimensional constraint either from the list or from within the drawing
- Customize the Quick Properties palette to display several constraint properties

When you enter the changes, the results will propagate across the drawing immediately.

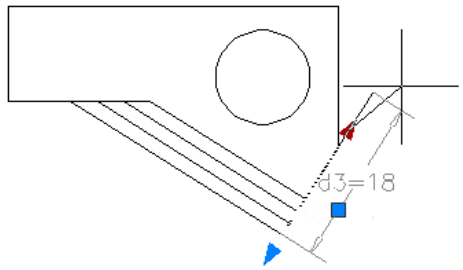
NOTE You cannot edit the Expression and Value properties for a reference constraint.

Modify Dimensional Constraints Using Their Grips

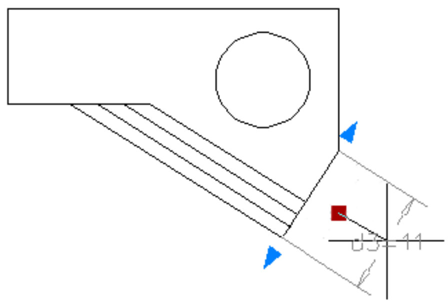
You can modify a constrained object either by using the triangular grips or the square grips on the associated dimensional constraint.

The triangular grips on dimensional constraints provide a way of changing the constraint value while maintaining the constraint.

For example, you can change the length of the diagonal line by using the triangular grips on the Aligned dimensional constraint. The diagonal line maintains its angle and the location of one of its endpoints.



The square grip on dimensional constraints provides a way of changing the location of the text and other elements.



Dynamic dimensional constraints are more limited than annotational dimensional constraints in where the text can be located.

NOTE Triangular grips are not available for dimensional constraints that reference other constraint variables in expressions.

For information about temporarily relaxing constraints, see Overview of Constraints on page 718.

To grip-edit a dimensional constraint

1 Select a constrained object.

2 Click the grips and drag to edit the geometry.

Toolbar: Parametric, Dimensional Constraints
Command entry: DIMCONSTRAINT

To edit a dimensional constraint in-place

1 Double-click a dimensional constraint to display the in-place text editor.

2 Enter the new name, value, or expression (name=value).

3 Press Enter to confirm the change.

Toolbar: Parametric, Dimensional Constraints
Command entry: DIMCONSTRAINT

To edit a dimensional constraint using the Properties palette

1 Select a dimensional constraint, right-click in the drawing area, and click Properties.

2 Enter the new values for Name, Expression, and Description text boxes.

Toolbar: Parametric, Dimensional Constraints
Command entry: DIMCONSTRAINT

To turn a dimensional constraint off

1 Click a constrained object in a drawing to select it.
The grips are displayed on the object to show that it is selected.

2 Move your cursor over a grip. The grip color turns red.

3 Click the grip.

4 Press and release the Ctrl key.

5 Move the object to the desired location.
The constraint is relaxed for the object, and you should be able to move it.

Toolbar: Parametric, Dimensional Constraints
Command entry: DIMCONSTRAINT

To edit the dimensions using the Parameters Manager palette

1 Click Parametric tab ➤ Manage panel ➤ Parameters Manager. 

2 Double-click the variable you want to edit.

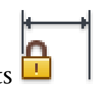
3 Press Tab to navigate across the columns.

4 Change the values in the appropriate column.

NOTE You can modify only the Name, Expression, and Description columns.

---

5 Press Enter.

Toolbar: Parametric, Dimensional Constraints 
Command entry: DIMCONSTRAINT

Quick Reference

Commands

DELCONSTRAINT

Removes all geometric and dimensional constraints from a selection set of objects.

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects.

LIST

Displays property data for selected objects.

PARAMETERS

Controls the associative parameters used in the drawing.

-PARAMETERS

Controls the associative parameters used in the drawing.

PARAMETERSCLOSE

Closes the Parameters Manager palette.

TEXTEDIT

Edits a dimensional constraint, dimension, or text object.

System Variables

CONSTRAINTNAMEFORMAT

Controls the text format for dimensional constraints.

CONSTRAINTRELAX

Indicates whether constraints are enforced or relaxed when editing an object.

CONSTRAINTSOLVEMODE

Controls constraint behavior when applying or editing constraints.

DIMCONSTRAINTICON

Displays the lock icon next to the text for dimensional constraints.

DYNCONSTRAINTDISPLAY

Displays or hides dynamic constraints.

DYNCONSTRAINTMODE

Displays hidden dimensional constraints when constrained objects are selected.

PARAMETERCOPYMODE

Controls how constraints and referenced variables are copied when replicating constrained geometry.

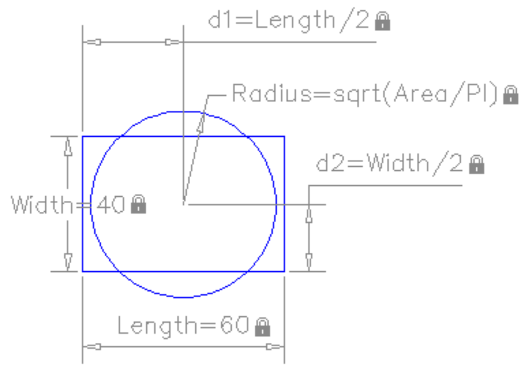
PARAMETERSSTATUS

Indicates whether the Parameters Manager palette is displayed or hidden.

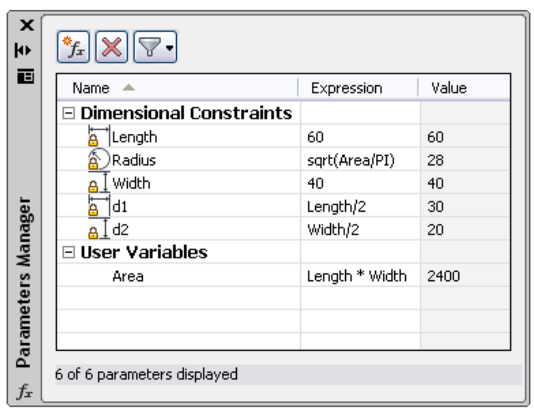
Constrain a Design with Formulas and Equations

You can control geometry using mathematical expressions that include the names of dimensional constraints, user variables, and functions.

Formulas and equations can be represented either as expressions within dimensional constraints or by defining a user variables. For example, the following illustration represents a design that constrains a circle to the center of the rectangle with an area equal to that of the rectangle.



The *Length* and *Width* dimensional constraints are set to constants. The *d1* and *d2* constraints are simple expressions that reference the *Length* and *Width*. The *Radius* dimensional constraint is set to an expression that includes the square root function, parentheses to determine the precedence of operations, the *Area* user variable, the division operator, and the constant, PI. These parameters are all displayed in the Parameters Manager.



As you can see, part of the equation for determining the area of the circle is included in the Radius dimensional constraint and part was defined as a user variable. Alternatively, the entire expression, sqrt (Length * Width / PI), could have been assigned to the *Radius* dimensional constraint, defined in a user variable, or some other combination.

Work with the Parameters Manager

The Parameters Manager displays dimensional constraints (both dynamic constraints and annotational constraints), reference constraints, and user variables. You can easily create, modify, and delete parameters from the Parameters Manager.

The Parameters Manager supports the following operations:

- Click the name of a dimensional constraint to highlight the constraint in the drawing.
- Double-click a name or expression to edit it.
- Right-click and click Delete to remove a dimensional constraint or user variable.
- Click a column heading to sort the list of parameters by name, expression, or value.

NOTE With imperial units, the Parameters Manager 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**".

Use Operators in Expressions

Dimensional constraints and user variables support the following operators within expressions:

Operator	Description
+	Addition
-	Subtraction or unary negation
%	Floating point modulo
*	Multiplication
/	Division
^	Exponentiation
( )	Parenthesis, expression delimiter
.	Decimal separator

Understand Precedence in Expressions

Expressions are evaluated according to the following standard mathematical rules of precedence:

1. Expressions in parentheses first, starting with the innermost set
2. Operators in standard order: unary negation first, then exponents, multiplication and division, and addition and subtraction last
3. Operators of equal precedence from left to right

Expressions are evaluated using the standard precedence as shown in the following table in descending order.

Functions Supported in Expressions

The following functions are available for use in expressions:

Function	Syntax
Cosine	cos(*expression*)
Sine	sin(*expression*)
Tangent	tan(*expression*)
Arc cosine	acos(*expression*)
Arc sine	asin(*expression*)
Arc tangent	atan(*expression*)
Hyperbolic cosine	cosh(*expression*)
Hyperbolic sine	sinh(*expression*)
Hyperbolic tangent	tanh(*expression*)
Arc hyperbolic cosine	acosh(*expression*)
Arc hyperbolic sine	asinh(*expression*)
Arc hyperbolic tangent	atanh(*expression*)
Square root	sqrt(*expression*)

Function	Syntax
Signum function (-1,0,1)	sign(*expression*)
Round to nearest integer	round(*expression*)
Truncate decimal	trunc(*expression*)
Round down	floor(*expression*)
Round up	ceil(*expression*)
Absolute value	abs(*expression*)
Largest element in array	max(*expression1*;*expression2*)
Smallest element in array	min(*expression1*;*expression2*)
Degrees to radians	d2r(*expression*)
Radians to degrees	r2d(*expression*)
Logarithm, base *e*	ln(*expression*)
Logarithm, base 10	log(*expression*)
Exponent, base e	exp(*expression*)
Exponent, base 10	exp10(*expression*)
Power function	pow(*expression1*;*expression2*)
Random decimal, 0-1	Random

In addition to these functions, the constants Pi and e are also available for use in expressions.

To create a user variable

1 Click Parametric tab ➤ Manage panel ➤ Parameters Manager.

2 Click the New User Parameter icon.

A new user variable is created under the User Variables group.

Toolbar: Parametric
Command entry: PARAMETERS

To reference a variable within an expression

1 Click Parametric tab ➤ Manage panel ➤ Parameters Manager.

2 Double-click the variable you want to reference.

3 Right-click the cell in the Name column, and click Copy.

4 Double-click the Expression column where you want to include the referenced variable.

5 Right-click the Expression column, and click Paste.

Toolbar: Parametric
Command entry: PARAMETERS

To include a function in an expression

1 Click Parametric tab ➤ Manage panel ➤ Parameters Manager.

2 Double-click the Expression column of the variable to which you want to add the function.

3 Right-click the Expression column, and click Expressions.

4 Select the function to insert it in the Expression column.

Toolbar: Parametric
Command entry: PARAMETERS

To modify a user parameter

1 Click Parametric tab ➤ Manage panel ➤ Parameters Manager.

2 Double-click the columns of the variable you want to edit.

3 Change the values of the appropriate columns.

NOTE You can modify only the Name, Expression, and Description columns.

4 Press Enter.

Toolbar: Parametric
Command entry: PARAMETERS

To delete a user parameter

1 Click Parametric tab ➤ Manage panel ➤ Parameters Manager.

2 Select the user variable you want to delete.

3 Click the Delete Parameter icon.

Toolbar: Parametric
Command entry: PARAMETERS

To select a constrained object associated with a user parameter

1 Click Parametric tab ➤ Manage panel ➤ Parameters Manager.

2 Click the dimensional variable to view the associated object in the drawing.

Toolbar: Parametric fx
Command entry: PARAMETERS

Quick Reference

Commands

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects.

PARAMETERS

Controls the associative parameters used in the drawing.

-PARAMETERS

Controls the associative parameters used in the drawing.

Extract Geometric Information from Objects

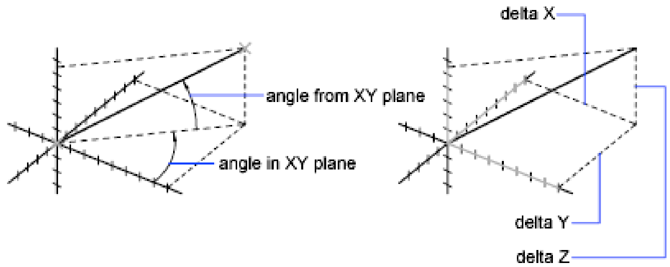
The inquiry and calculation commands can provide information about objects in your drawing and do useful calculations.

Obtain Distances, Angles, and Point Locations

You can obtain information about the relation between two specified points or multiple points; for example, the distance between points or their angle in the *XY* plane.

To determine the relation between points, you can display the

- Distance between them
- Angle between the points in the *XY* plane
- Angle of the points from the *XY* plane
- Delta, or changed, *X*, *Y*, and *Z* distances between them



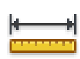
The ID command lists the *X*, *Y*, and *Z* coordinate values of a specified point.

See also:

- Overview of Coordinate Entry on page 629

To calculate the distance and angle between two points

1. Click Tools tab ➤ Inquiry panel ➤ Distance.
2. Specify a first and second point for the distance you want to calculate.
3. Press ENTER.
   The distance displays at the Command prompt in the current units format.

Toolbar: Inquiry 
Command entry: DIST

To calculate the distance and angle between multiple points

1. Click Tools tab ➤ Inquiry panel ➤ Distance.
2. For the distance you want to calculate, specify a first and second point.
3. For the distance you want to calculate, specify the next point or points.
4. Press ENTER.
   The distance displays at the Command prompt in the current units format.

Toolbar: Inquiry
Command entry: DIST

Quick Reference

Commands

DIST

Measures the distance and angle between two points.

ID

Displays the UCS coordinate values of a specified location.

MEASUREGEOM

Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

System Variables

DISTANCE

Stores the distance computed by the DIST command.

Utilities

No entries

Command Modifiers

No entries

Obtain Area and Mass Properties Information

You can obtain the area, perimeter, and mass properties defined by selected objects or a sequence of points.

You can calculate the area and perimeter of a sequence of points. You can also obtain the area, perimeter, and mass properties of any of several types of objects.

TIP A fast way to calculate an area bounded by several objects in 2D is to use the BOUNDARY command. With BOUNDARY, you can pick a point within the area to create a closed polyline or region. You can then use the Properties palette or the LIST command to find the area and perimeter of the polyline or region.

Use Commands to Calculate Area

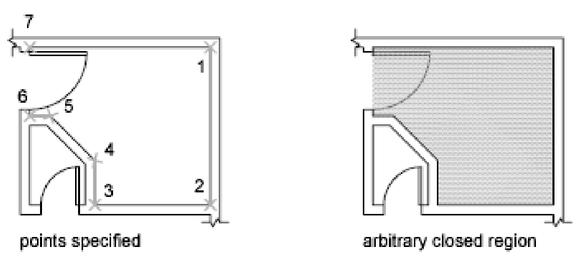
With the MEASUREGEOM and AREA commands, you can specify a series of points or select an object to calculate area. If you need to calculate the combined area of mutiple objects, you can keep a running total as you add or subtract one area at a time from the selection set. You cannot use window selection or crossing selection to select objects.

Total area and perimeter are saved in the AREA and PERIMETER system variables.

In addition to area, with the MEASUREGEOM command, you can obtain geometric information from objects such as distance, radius, angle, and volume.

Define an Area

You can measure an arbitrary closed region defined by the points you specify. The points must lie on a plane parallel to the *XY* plane of the current UCS.



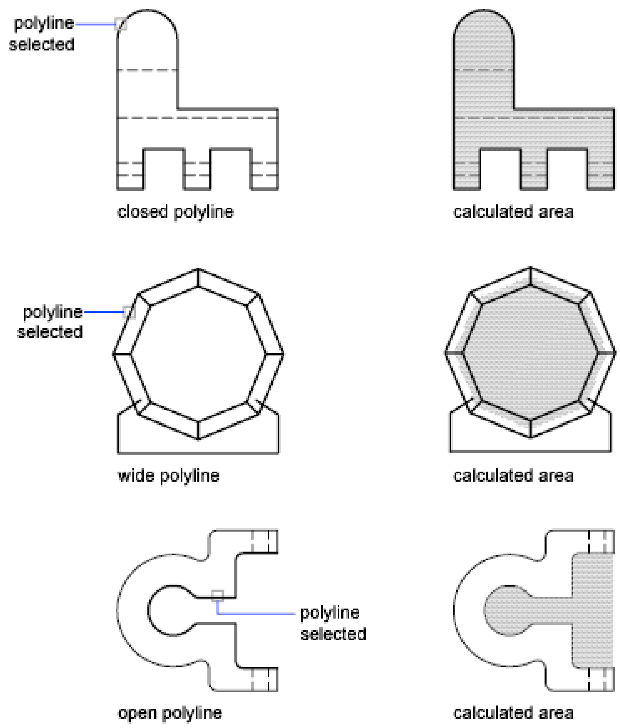
Calculate the Area, Perimeter, or Circumference of an Object

You can calculate the enclosed area and perimeter or circumference of circles, ellipses, polylines, polygons, regions, and AutoCAD 3D solids. The information displayed depends on the type of object selected:

- **Circles.** Area and circumference display.
- **Ellipses, closed polylines, polygons, planar closed spline curves, and regions.** Area and perimeter display. For wide polylines, this area is defined by the center of the width.

- **Open objects such as open spline curves and open polylines.** Area and length display. Area is calcuated as though a straight line connects the start point and endpoint.
- **AutoCAD 3D solids.** Total 3D area for the object displays.

Example: How Various Areas Are Calculated



Combined Areas

Calculate Combined Areas

You can calculate the total area of multiple areas by specifying points or by selecting objects. For example, you can measure the total area of selected rooms in a floor plan.

Subtract Areas from Combined Areas

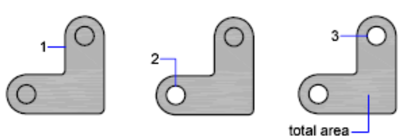
You can subtract more than one area from a combined area as you calculate. For example, if you have calculated the area of a floor plan, you can subtract the area of a room.

Example: Subtraction of Areas from a Calculation

In the following example, the closed polyline represents a metal plate with two large holes. The area of the polyline is first calculated and then the area of each hole is subtracted. The area and perimeter or circumference of each object displays, with a running total after each step.

The command prompt sequence is

Command: **area**
Specify first corner point or [Object/Add/Subtract]: **a**
Specify first corner point or [Object/Subtract]: **o**
(ADD mode) Select objects: *Select the polyline (1)*
Area = 0.34, Perimeter = 2.71
Total area = 0.34
(ADD mode) Select objects: *Press* ENTER
Specify first corner point or [Object/Subtract]: **s**
Specify first corner point or [Object/Add]: **o**
(SUBTRACT mode) Select objects: *Select the lower circle (2)*
Area = 0.02, Circumference = 0.46
Total area = 0.32
(SUBTRACT mode) Select objects: *Select the upper circle (3)*
Area = 0.02, Circumference = 0.46
Total area = 0.30
(SUBTRACT mode) Select circle or polyline: *Press* ENTER
Specify first corner point or [Object/Add]: *Press* ENTER



You can also use REGION to convert the plate and the holes to regions, subtract the holes, and then use the Properties palette or the LIST command to find the area of the plate.

TIP Use the QuickCalc calculator to convert from one system of area units to another. For more information, see Use the QuickCalc Calculator on page 773.

Calculate Mass Properties

With the MASSPROP command, you can analyze 3D solids and 2D regions for their mass properties including volume, area, moments of inertia, center of gravity, and so on. In addition, the result of the computations can be saved to a text file.

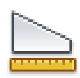
See also:

- Create and Combine Areas (Regions) on page 845
- Overview of Object Properties on page 523

To calculate an area you define

1. Click Tools tab ➤ Inquiry panel ➤ Area.
2. Specify points in a sequence that define the perimeter of the area you want to measure. Press ENTER.

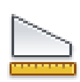
   The first and last points connect to form a closed area. The area and perimeter display in the current units format.

Toolbar: Inquiry 
Command entry: AREA

To calculate the area of an object

1. Click Tools tab ➤ Inquiry panel ➤ Area.
2. At the command prompt, enter **o** (Object).
3. Select an object.

   The area and perimeter of the selected object display in the current units format.

Toolbar: Inquiry 
Command entry: AREA

To add an area as you calculate

1. Click Tools tab ➤ Inquiry panel ➤ Area.

2 Enter **a** (Add).

3 Do one of the following:

- Specify points to define the area you want to add. Press ENTER.
- Enter **o** (Object) and select the objects you want to add.
  Each new area and a running total of all areas displays in the current units format.

4 Press ENTER twice to end the command.

Toolbar: Inquiry
Command entry: AREA

To subtract an area as you calculate

1 While a combined area displays, enter **s** (Subtract).

2 Do one of the following:

- Specify points to define the area you want to subtract. Press ENTER.
- Enter **o** (Object) and select the objects you want to subtract.

The running total of all areas updates and displays as you define new areas.

3 Press ENTER to end the command.

Toolbar: Inquiry
Command entry: AREA

Quick Reference

Commands

AREA

Stores the last area computed by the AREA command.

LIST

Displays property data for selected objects.

MASSPROP

Calculates the mass properties of regions or 3D solids.

MEASUREGEOM

Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

PROPERTIES

Controls properties of existing objects.

QUICKCALC

Opens the QuickCalc calculator.

UNITS

Controls coordinate and angle display formats and precision.

System Variables

AREA

Stores the last area computed by the AREA command.

PERIMETER

Stores the last perimeter value computed by the AREA or LIST command.

Utilities

No entries

Command Modifiers

No entries

Use a Calculator

You can access a calculator function as you work with the program. You can use either the QuickCalc calculator interface or the command prompt calculator.

Use the QuickCalc Calculator

With the QuickCalc calculator, an interface that looks and functions like a hand-held calculator, you can perform mathematical, scientific, and geometric calculations, convert units of measurement, manipulate the properties of objects, and evaluate expressions.

Overview of QuickCalc

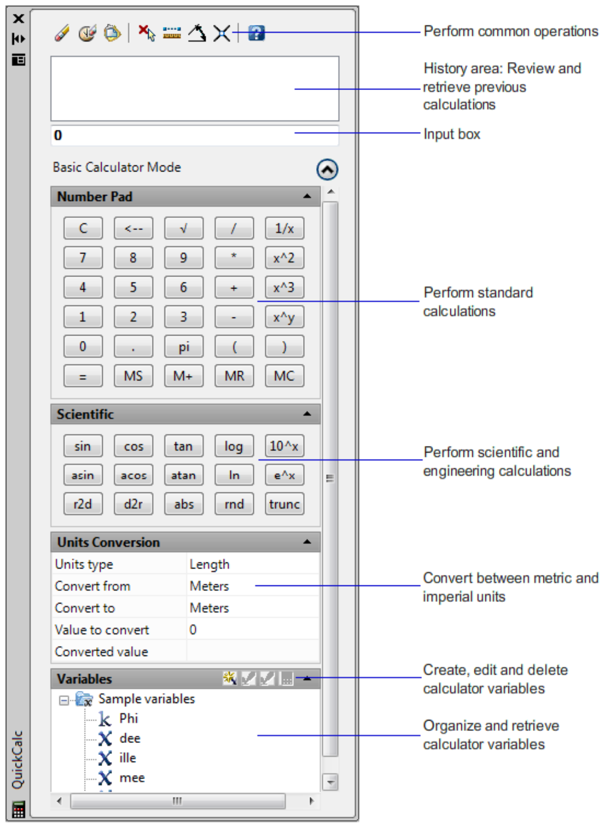
The QuickCalc calculator includes basic features similar to most standard mathematical calculators.

In addition, QuickCalc has features specific to AutoCAD such as geometric functions, a Units Conversion area, and a Variables area.

Unlike most calculators, QuickCalc is an expression builder. For greater flexibility, it does not immediately calculate an answer when you click a function. Instead, you compose an expression that you can easily edit and, when you are finished, you click the equal sign (=) or press ENTER. Later, you can retrieve the expression from the History area, modify it, and recalculate the results.

With QuickCalc, you can

- Perform mathematical and trigonometric calculations
- Access and review previously entered calculations for re-evaluation
- Use the calculator with the Properties palette to modify object properties
- Convert units of measurement
- Perform geometric calculations related to specific objects
- Copy and paste values and expressions to and from the Properties palette and the command prompt
- Perform computations on mixed numbers (fractions), and feet and inches
- Define, store, and use calculator variables
- Use geometric functions from the CAL command



Change QuickCalc Size and Appearance

Click the More/Less button on the calculator and only the Input box and History area are displayed. You can use the expand/collapse arrows to open and close areas. You can also control the size, location, and appearance of QuickCalc. See Set Interface Options on page 111.

To clear the Input box

Do one of the following:

- Click View tab ➤ Palettes panel ➤ QuickCalc. 
- On the QuickCalc toolbar, click the Clear button.
- On the QuickCalc Number Pad, click the Clear button.

Any current value or expression in the Input box is cleared and the value is reset to 0.

Command entry: QUICKCALC

To use basic math functions in QuickCalc

1. Click View tab ➤ Palettes panel ➤ QuickCalc. 
2. On the Number Pad, click a number.
3. Click an operator (+, -, *, /) button. Then enter the next number, and so on.
4. Click the equal (=) sign.

   The result is displayed in the Input box. The expression and result are also displayed in the History area.

Command entry: QUICKCALC

To use scientific functions in QuickCalc

1. Click View tab ➤ Palettes panel ➤ QuickCalc. 
2. On the Number Pad, enter a value.
3. In the Scientific area, click a function.
4. On the Number Pad, click the equal (=) sign.

   The result is displayed in the Input box.

Command entry: QUICKCALC

To get the absolute value of a number using QuickCalc

1. Click View tab ➤ Palettes panel ➤ QuickCalc. 
2. With the current value displayed, in the Scientific area click the abs button.
3. On the Number Pad, click the equal (=) sign.

The absolute value is displayed in the Input box.

Command entry: QUICKCALC

To round a number to the nearest integer using QuickCalc

1 Click View tab ➤ Palettes panel ➤ QuickCalc. 

2 With the current value displayed, in the Scientific area click the rnd button.

3 On the Number Pad, click the equal (=) sign.

   The number, rounded to the nearest integer, is displayed in the Input box.

Command entry: QUICKCALC

To display only the integer portion of a number with QuickCalc

1 Click View tab ➤ Palettes panel ➤ QuickCalc. 

2 With the current value displayed, in the Scientific area click the trunc button.

3 On the Number Pad, click the equal (=) sign.

   The integer portion of the number is displayed in the Input box without the decimal portion.

Command entry: QUICKCALC

To store a value in the QuickCalc memory

- Click View tab ➤ Palettes panel ➤ QuickCalc. 
- With the current value displayed, click MS.
  The previous value is overwritten and the new value is stored in memory.

Command entry: QUICKCALC

To add the current value to the value stored in the QuickCalc memory

- Click View tab ➤ Palettes panel ➤ QuickCalc.
- With the current value displayed, click M+.
  The sum of the new value and the value already in memory are displayed.

Command entry: QUICKCALC

To restore the value stored north QuickCalc memory

- Click View tab ➤ Palettes panel ➤ QuickCalc.
- With the current value displayed, click MR.
  The displayed value or expression is appended and the value stored in memory is displayed.

Command entry: QUICKCALC

To clear the value stored in the QuickCalc memory

- Click View tab ➤ Palettes panel ➤ QuickCalc.
- Click MC.
  The value currently stored in memory is cleared.

Command entry: QUICKCALC

Quick Reference

Commands

QUICKCALC

Opens the QuickCalc calculator.

QCCLOSE

Closes QuickCalc.

System Variables

QCSTATE

Indicates whether the QuickCalc calculator is open or closed.

Utilities

No entries

Command Modifiers

No entries

Access QuickCalc and Understand Its Behavior

Use QuickCalc directly as you would with a desktop calculator, or use it transparently within a command or the Properties palette.

There are three ways to work with QuickCalc within the program:

- Directly from the Tools menu, the Standard toolbar, a shortcut menu, or the command prompt
- Transparently during a command from a shortcut menu or the command prompt
- Transparently from the Properties palette

The method that you choose depends on how you are using QuickCalc.

Use QuickCalc Directly

When you work directly with QuickCalc, you can perform calculations and unit conversions just as you would with a desktop calculator. You can use the Windows clipboard (CTRL+C, CTRL+V) to transfer the results of your calculations to other parts of the program or to external programs. Calculations that you perform directly do not affect or change anything in your drawing.

You can access QuickCalc directly in the following ways:

- Click Tools menu ➤ Palettes ➤ QuickCalc.
- On the Standard toolbar, click the QuickCalc button.
- In the drawing editor (with no command active), right-click and click QuickCalc.

- At the command prompt, enter **quickcalc**.
- At the command prompt, enter **qc**.

Use QuickCalc Transparently from Within a Command

During a command, you can access QuickCalc transparently in the following ways:

- Right-click to display the shortcut menu. Click QuickCalc.
- At the command prompt, enter **quickcalc**.
- At the command prompt, enter **qc**.

Calculations that you transfer to the command prompt affect the drawing. For example, if you are drawing a line with the LINE command and you pass an expression to the command prompt from the calculator, the next point of the line uses the results, which might be a distance or coordinate value. In QuickCalc, you click the Apply button to pass the value to the drawing based on that expression.

NOTE When using QuickCalc transparently to calculate a value for direct distance entry, Apply transfers the value to the command prompt. To use the value, position the crosshairs to determine the direction and then press ENTER.

Use QuickCalc Transparently from the Properties Palette

When you want to modify properties and apply evaluated expressions to objects in your drawing, you can access QuickCalc transparently from the Properties palette. Use the following method:

- Click any box in the Properties palette that contains a numeric value. Then click the QuickCalc button that appears in the box.
- After calculating a value, transfer the result to the Properties palette by clicking the Apply button.

The object or drawing property is modified and the changes are displayed in the drawing.

Calculate Mathematical Expressions in a Dialog Box

You can also enter and evaluate mathematical expressions in a dialog box using the following format: *=expression<ALT+ENTER>*.

NOTE To evaluate expressions in a dialog box, make sure the system variable, CALCINPUT, is set to 1.

To use the QuickCalc calculator

Do *one* of the following:

- Click View tab ➤ Palettes panel ➤ QuickCalc. 
- QuickCalc

  
- Right-click the drawing area to display a shortcut menu. Click QuickCalc.

Command entry: QUICKCALC

To use the QuickCalc calculator within a command

Do *one* of the following:

- At the command prompt, enter '**quickcalc** or '**qc**.
- Right-click to display a shortcut menu. Click QuickCalc.

To use the QuickCalc calculator with the Properties palette

1. Click View tab ➤ Palettes panel ➤ Properties. 
2. Open the Properties palette.
3. Click one or more objects.
4. Click a box with a numeric property.
5. Click the QuickCalc button in the box.

Command entry: QUICKCALC

Quick Reference

Commands

QUICKCALC

Opens the QuickCalc calculator.

QCCLOSE

Closes QuickCalc.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Enter, Evaluate, and Retrieve Expressions

Enter expressions in QuickCalc using standard mathematical rules of precedence; review and retrieve computations from the History area; and understand the rules for using imperial units: length, area, and volume.

QuickCalc evaluates expressions according to the following 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

The Input box of the calculator is where you enter and retrieve expressions. With QuickCalc, there are two ways you can enter data in the Input box. You can either enter expressions using the QuickCalc Number Pad buttons, or you can use the computer keyboard or numeric keypad. To use the computer numeric keypad, you must have NUMLOCK on.

To evaluate an expression, click the equal (=) sign on the QuickCalc Number Pad, or press ENTER on the computer keyboard.

Understand the Syntax of Expressions

The syntax for QuickCalc expressions and command prompt calculator expressions is identical. For example, to perform an operation on the vector or coordinates 5,2,0, you enter [5,2,0] in the Input box.

You can use the GETVAR function to read the value of a system variable. The syntax is

getvar(*variable_name*)

For more information see CAL.

Use the History Area

The History area keeps an ongoing record of calculations, similar to the paper tape in a physical desk calculator. You can use the History area to review previous operations and pass them back to the Input box for re-evaluation with different parameters.

Rules for Displaying and Handling Units

QuickCalc adheres to the following rules:

- Results of calculations are always expressed in decimal format unless a distance is entered in feet and inches
- Angular values entered in the Input box are assumed to be degrees regardless of the settings in the Drawing Units dialog box. To specify radians, grads, and degrees, append an **r**, **g,** or **d** after the angle value.
- Results of angular calculations are always expressed in degrees with full AutoCAD precision.

When the drawing units are set to architectural units, the calculator displays the results of calculations of imperial units in the architectural format and rounds to the display precision (LUPREC) specified in the drawing. The results for all other calculations display in decimal format with full precision.

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, QuickCalc 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"**.

You can use QuickCalc to calculate square feet and cubic feet. To enter square or cubic feet, you must enter units using these abbreviations:

- **sq. ft. or sq ft**
- **cu. ft. or cu ft**

Convert Decimal Units into Imperial Units

For distance measurements, enter an inches sign (") after the number in the results display. For example, if the computed distance is 15, enter " after the 15, press ENTER or click = and the result displays in imperial units as 1'-3".

For computed results, enter the initial values in feet (') and inches (") to display the results in feet and inches. For example:

- 5 * 6 = 30
- 5" * 6 = 2'-6"
- 5" * 6" = 30 sq. in.
- 5" * 0'-6" = 0.208333333 sq. ft.

To copy and paste a value from the QuickCalc Input box

1. Right-click the value in the Input box. Click Copy.
   The current entry is copied to the clipboard.
2. Click in the new location and then right-click. Click Paste.

The value is copied to the new location.

To paste a value from the QuickCalc Input box to the command line

- On the QuickCalc toolbar, click the Paste Value to Command Line button. The value in the Input box is pasted into the command line.

To clear the History area

- On the QuickCalc toolbar, click the Clear History button.

Shortcut menu: Right-click the History area. Click Clear History.

To reuse a stored value or expression in the QuickCalc History area

1. Click the C button to clear the Input box if needed.
2. In the History area, double-click a value or expression. The value or expression is displayed in the Input box.

   NOTE The cursor must be on the value or expression to select it.

To change the font color of values or expressions in the QuickCalc History area

1. Right-click in the History area. Click either Value Font Color or Expression Font Color.
2. In the Color dialog box, click a Basic Color or click Define Custom Colors.

   With Define Custom Colors, you can select a custom color and add it to Custom Colors.
3. Click OK.

   The colors you selected for values and expressions in the History area are displayed.

   Right-click Expression Font Color or Value Font Color. Click Basic Color or Define Custom Colors.

To copy and paste an expression from the QuickCalc History area

1. Right-click the expression in the History area. Click Copy.

   The current entry is copied to the clipboard.

2 Right-click in the new location. Click Paste.

The expression is copied to the new location.

To append a value or expression from the QuickCalc History area to the Input box

- Double-click the value or expression in the History area.
  The value or expression is appended to the Input box.

  NOTE The cursor must be on the value or expression to select it.

Shortcut menu: Right-click Append Value to Input Area or Append Expression to Input Area.

To modify a property in the Properties palette with QuickCalc

1 Select an object.

NOTE If QuickCalc is displayed on your desktop when you work with the Properties palette, it is temporarily hidden while you use the modal calculator from within the Properties palette.

2 In the Geometry section of the Properties palette, click on the value of a property.

A small calculator icon is displayed to the right of the value.

NOTE Only properties that are displayed with a white background can be changed.

3 Click the calculator icon.

QuickCalc opens and displays the current value of the object in the Input box.

4 Perform a calculation on the displayed value and click the equal sign (=) button.

The new value is displayed in the Input box.

5 Click Apply.

NOTE The Apply button is only available for editable number-based properties.

The calculator closes and the new value is displayed in the Properties palette. The object is modified in the drawing.

To obtain X, Y, Z coordinate values for a point using QuickCalc

1 **NOTE** The QuickCalc toolbar Get Coordinates button uses the cur function.

On the QuickCalc toolbar, click the Get Coordinates button.

QuickCalc temporarily closes and you are prompted to specify a point.

2 In the drawing, click a point.

QuickCalc opens and displays the coordinate values of the point in the Input box.

To measure the distance between two points using QuickCalc

1 **NOTE** The QuickCalc toolbar Distance Between Two Points button uses the dist(p1,p2) function.

On the QuickCalc toolbar, click the Distance Between Two Points button.

QuickCalc temporarily closes and you are prompted to specify two points.

2 In the drawing, click the first point and then the second point.

QuickCalc opens and displays the value of the distance between the two points in the Input box.

To get the angle of a line defined by two points using QuickCalc

1 **NOTE** The QuickCalc toolbar Angle of Line Defined by Two Points button uses the ang(p1,p2) function.

On the QuickCalc toolbar, click the Angle of Line Defined by Two Points button.

QuickCalc temporarily closes and you are prompted to specify two points.

2 Enter the coordinate values for the first point and then the second point.

QuickCalc opens and appends the value of the angle between the two points to the end of any value or expression already present in the Input box.

To get the intersection of a line defined by four points using QuickCalc

1 **NOTE** The QuickCalc Intersection of Two Lines Defined by Four Points button uses the ill(p1,p2,p3,p4) function.

   On the QuickCalc toolbar, click the Intersection of Two Lines Defined by Four Points button.

   QuickCalc temporarily closes and you are prompted to specify four points.

2 Enter the coordinate values for the first point of line one, then the second point of line one. Next, enter the coordinate values for the first point of line two, then the second point of line two.

   QuickCalc opens and appends the value of the evaluated expression to the end of any value or expression already present in the Input box.

Quick Reference

Commands

QUICKCALC

Opens the QuickCalc calculator.

QCCLOSE

Closes QuickCalc.

System Variables

No entries

Utilities

No entries

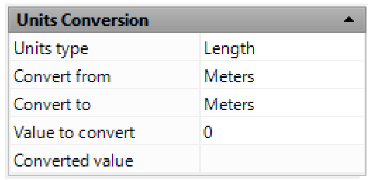
Command Modifiers

No entries

Convert Units of Measurement

In the Units Conversion area of QuickCalc, you can obtain equivalent values for different units of measurement.

Unit conversions are available for length, area, volume, and angular values. Based on which unit type you select, you can then select a list of units to convert from and a list of units to convert to.



The Value to Convert box automatically displays the value from the Input box. You can also enter a different value. The results of the units conversion displays in the Converted Value box. You can paste this result to the Input box by clicking the QuickCalc icon in the Converted Value box.

NOTE In the Value to Convert box, enter decimal values without units.

To convert units of measurement with QuickCalc

1. In the Units Conversion area, select a unit category on the Units Type list.
2. In the Convert From list, select the type of unit you are converting from.
3. In the Convert To list, select the type of unit you are converting to.
4. In the Value to Convert box, enter the value you want to convert. Press ENTER.

   The converted value is displayed in the Converted Value box.

To copy a unit conversion result to the Input box of QuickCalc

- On the Units Conversion title bar, click the Return Conversion Value to Input Area button.
  The converted value is displayed in the Input box.

To convert radians to degrees using QuickCalc

1. On the Number Pad, enter a value in radians.
2. In the Scientific area, click the r2d button.
3. On the Number Pad, click the equal (=) sign.

   The conversion is displayed in the Input box.

To convert degrees to radians using QuickCalc

1 On the Number Pad, enter a value in degrees.
2 In the Scientific area, click the d2r button.
3 On the Number Pad, click the equal (=) sign.
   The conversion is displayed in the Input box.

Quick Reference

Commands

QUICKCALC

Opens the QuickCalc calculator.

QCCLOSE

Closes QuickCalc.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Create and Use Calculator Variables

The Variables area of QuickCalc stores calculator variables that you can access as needed. Calculator variables can either be constants or functions.

You can use the Variables area to define, store, and retrieve *calculator variables*. Calculator variables can either be constants (coordinates/vectors, real numbers, and integers) or functions. In the Variables area, you can

- Click a calculator variable to display information such as value, type, and description in the Details box at the bottom of the Variables area.
- Double-click a calculator variable to load it into the QuickCalc Input box.

Additional operations are available on the shortcut menus in the Variables area.

Create New Calculator Variables

You can create new calculator variables using the shortcut menus in the Variables area. When defining new calculator variables in the Variable Definition dialog box, the following rules apply:

- **Constants.** Any expression entered in the Value or Expression text entry box is evaluated before the calculator variable is stored. Calculator variables that are defined as constants are available "globally." You can access and use global constants in different drawings and sessions.
- **Functions.** Any expression entered in the Value or Expression text entry box is stored as text. Functions are evaluated when used in the QuickCalc Input box.

Create Global Constants

You can use one of the following methods to create global constants:

- Enter an expression in the Input box using the format **$***variable_name***=***value*. For example, to define the golden ratio to 8 decimal places as a global constant called Phi, enter $Phi=1.61803399 in the Input box.
- Click the New Variable button in the Variables area title bar. In the Variables Definition dialog box, click Constant and fill in the other boxes.
- Right-click the Variables area. Click New Variable.

Access Global Constants

You can access global constants and pass them to the Input box of the QuickCalc as follows:

- Double-click a variable in the Variables area of QuickCalc.
- Click a variable from the list of calculator variables, and click the Return Variable to Input Area button.
- Enter a dollar sign ($) followed by the variable name, and press ENTER.

To use a global constant in a text or numeric entry box in a window or dialog box, use the syntax: **=$***variable_name* followed by pressing the END key. For example, to use the previously mentioned global variable, Phi, enter **=$Phi** and press the END key.

NOTE In QuickCalc, only constants can be directly referred to by their global variable names in the text or numeric entry boxes in windows and dialog boxes.

Use Shortcut Functions

Several sample calculator variables have been predefined and stored in the Shortcut Functions category. These are geometric expressions that combine CAL functions with the Endpoint Snap mode. The following table describes the predefined variables that are available in the Variables area of the calculator.

Variable	Shortcut For	Description
dee	dist(end,end)	Distance between two endpoints
ille	ill(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	vee(end,end)	Vector from two endpoints
vee1	vec1(end,end)	Unit vector from two endpoints

You can easily modify these calculator variables or create your own. For more information, see the CAL command.

Organize Variables into Categories

You can organize calculator variables in the Variables area under several categories. This results in a one-level tree structure. The Shortcut Functions category has already been created and contains several functions.

Use the shortcut menu in the Variables area to create, rename, or delete variable categories.

To use a predefined variable in a QuickCalc expression

1. In the Variables area, click the variable you want to use.
2. On the QuickCalc Variables title bar, click the Return Variable to Input Area button.

   The variable is displayed in the Input box as part of your expression.

To create a new variable in QuickCalc

1. On the QuickCalc Variables title bar, click the New Variable button.
2. In the Variable Definition dialog box, under Variable Type, select Constant or Function.
3. In the Variable Definition dialog box, under Variable Properties Name, enter a name for the variable. Variable names cannot contain spaces or special characters.

   NOTE When referencing this variable from the Input box, the variable name must begin with the dollar sign ($) to differentiate it from local LISP variables.

   ---

4. Under Variable Properties Group With, click New.
5. In the Category Definition dialog box, under Category Properties Name, enter a name for the new category.
6. Under Description, enter a description for the new category. Click OK.
7. In the Variable Definition dialog box, under Value or Expression, enter a value or expression for the new variable.
8. Under Description, enter a description of the new variable. Click OK.

   The new variable is now displayed in the Variables area.

To edit a variable in QuickCalc

1. In the Variables area, click the variable you want to edit.
2. On the QuickCalc Variables title bar, click the Edit Variable button.
3. In the Variable Definition dialog box, make the edits to the variable. Click OK.

To delete a variable in QuickCalc

1. In the Variables area, click the variable you want to delete.

2 On the QuickCalc Variables title bar, click the Delete button.

To create a new global constant in the Input box of QuickCalc

- In the Input box of QuickCalc, enter the following syntax:
  $*variable_name*=*value*.
  For example, you could center **$Phi=1.618**

  NOTE Global variables are not case-sensitive.

  QuickCalc adds the global constant to the list of variables in the Variables area.

To access a global constant from dialog box or window

- In any text or numeric entry box, enter an expression using the syntax: =**$***variable_name* followed by pressing the END key.

To evaluate mathematical expressions in a dialog box

1 If necessary, set the CALCINPUT system variable to 1.

2 In a dialog box, in an area where you can enter a numeric value, enter a mathematical expression in the following format: =*expression* (for example, =5+3).

3 Press Alt+Enter on the keyboard.

The expression is evaluated and its value is displayed.

CALCINPUT

Quick Reference

Commands

QCCLOSE

Closes QuickCalc.

QUICKCALC

Opens the QuickCalc calculator.

System Variables

CALCINPUT

Controls whether mathematical expressions and global constants are evaluated in text and numeric entry boxes of windows and dialog boxes.

Utilities

No entries

Command Modifiers

No entries

Use the Command Prompt Calculator

By entering an expression in the command prompt calculator, you can quickly solve a mathematical problem or locate points in your drawing.

The CAL command runs the 3D calculator utility to evaluate vector expressions (combining points, vectors, and numbers) and real and integer expressions. The calculator performs standard mathematical functions. It also contains a set of specialized functions for calculations involving points, vectors, and AutoCAD geometry. With the CAL command, you can

- Calculate a vector from two points, the length of a vector, a normal vector (perpendicular to the *XY* plane), or a point on a line
- Calculate a distance, radius, or angle
- Specify a point with the pointing device
- Specify the last-specified point or intersection
- Use object snaps as variables in an expression
- Convert points between a UCS and the WCS
- Filter the *X*, *Y*, and *Z* components of a vector
- Rotate a point around an axis

Evaluating Expressions

CAL evaluates expressions according to standard mathematical rules of precedence.

Mathematical operators in order of precedence

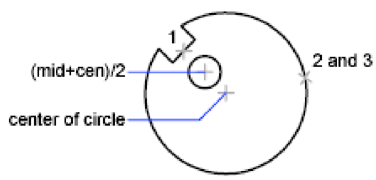
Operator	Operation
( )	Groups expressions
^	Indicates numeric exponent
*, /	Multiplies and divides numbers
+, -	Adds and subtracts numbers

Calculating Points

You can use CAL whenever you need to calculate a point or a number within a command.

For example, you enter **(mid+cen)/2** to specify a point halfway between the midpoint of a line and the center of a circle.

The following example uses CAL as a construction tool. It locates a center point for a new circle, and then calculates one fifth of the radius of an existing circle.



Here is the command prompr sequence:

Command: **circle**
Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: '**cal**
>> Expression: **(mid+cen)/2**
>> Select entity for MID snap: *Select the notch line (1)*
>> Select entity for CEN snap: *Select the large circle (2)*
Diameter/<Radius>: '**cal**
>> Expression: **1/5*rad**
>> Select circle, arc or polyline segment for RAD function: *Select the large circle (3)*

Calculate Mathematical Expressions in a Dialog Box

You can also enter and evaluate mathematical expressions in a dialog box using the following format: *=expression<ALT+ENTER>*.

NOTE To evaluate expressions in a dialog box, make sure the system variable, CALCINPUT, is set to 1.

To start the command prompt calculator

Do *one* of the following:

- At the command prompt, enter CAL. Then, enter a CAL expression.
- At a prompt for a command in progress, enter 'CAL to start the CAL command transparently. Then, enter a CAL expression to calculate a value for that prompt.

Quick Reference

Commands

CAL

Evaluates mathematical and geometric expressions.

System Variables

CALCINPUT

Controls whether mathematical expressions and global constants are evaluated in text and numeric entry boxes of windows and dialog boxes.

Utilities

No entries

Command Modifiers

No entries

Draw Geometric Objects

17

You can create a range of objects, from simple lines and circles to spline curves, and ellipses. In general, you draw objects by specifying points with the pointing device or by entering coordinate values at the command prompt.

Draw Linear Objects

A line, the most basic object, can be one segment or a series of connected segments.

Draw Lines

You can close a sequence of line segments so that the first and last segments are joined.

You can assign properties to lines including color, linetype, and lineweight. For more information about properties, see Work with Object Properties on page 523.

You specify the locations that define the endpoints of each line with precision. You can

- Enter the coordinate values for an endpoint, using either absolute or relative coordinates
- Specify an object snap relative to an existing object. For example, you can specify the center of a circle as one endpoint of the line
- Turn grid snap on and snap to a location

There are other methods for creating precise lines. A highly efficient technique is to offset a line from an existing line, and then trim or extend it to the desired length.

Use polyline objects instead of line objects if you want the segments to be connected as a single object.

See also:

- Use Coordinates and Coordinate Systems (UCS) on page 629
- Use Object Snaps on page 676
- Adjust Grid and Grid Snap on page 686
- Draw Polylines on page 801
- Offset an Object on page 1111
- Break and Join Objects on page 1138

To draw lines

1 Click Home tab ➤ Draw panel ➤ Line.

2 Specify the start point.

  You can use the pointing device or enter coordinate values at the command prompt.

3 Complete the first line segment by specifying the endpoint.

  To undo the previous line segment during the LINE command, enter **u** or click Undo on the toolbar.

4 Specify the endpoints of any additional line segments.

5 Press ENTER to end or **c** to close a series of line segments.

  To start a new line at the endpoint of the last line drawn, start the LINE command again and press ENTER at the Specify Start Point prompt.

Toolbar: Draw
Command entry: LINE

Quick Reference

Commands

LINE

Creates straight line segments.

RAY

Creates a line that starts at a point and continues to infinity.

XLINE

Creates a line of infinite length.

System Variables

No entries

Utilities

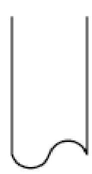
No entries

Command Modifiers

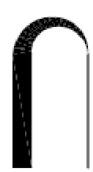
No entries

Draw Polylines

A polyline is a connected sequence of segments created as a single object. You can create straight line segments, arc segments, or a combination of the two.



pipe symbol



differing widths

Polylines are ideal for applications including the following:

- Contour lines for topographic, isobaric, and other scientific applications
- Wiring diagrams and printed cicuit board layouts
- Process and piping diagrams

■ Extrusion profiles and extrusion paths for 3D solid modeling

Polylines can be created with several commands including PLINE, RECTANG, POLYGON, DONUT, BOUNDARY, and REVCLOUD. All of these commands result in a LWPOLYLINE (lightweight polyline) object type.

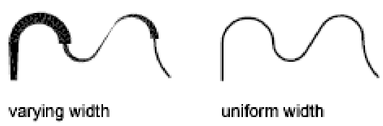
With the 3DPOLY command, you can create non-planar polylines that result in a POLYLINE object type. Fewer options are available with 3D polylines.

After you create a polyline, you can edit it using grips or PEDIT. You can use EXPLODE to convert polylines to individual line and arc segments.

NOTE You can convert a spline-fit polyline created with PEDIT into a true spline object with SPLINE.

Create Wide Polylines

You can draw polylines of various widths by using the Width and Halfwidth options. You can set the width of individual segments and make them taper gradually from one width to another. These options become available after you specify a starting point for the polyline.



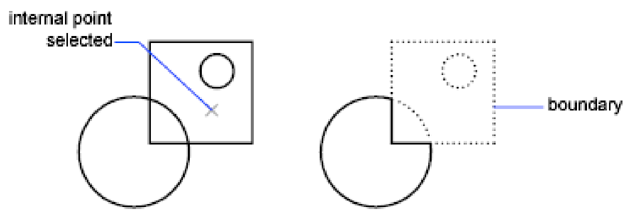
The Width and Halfwidth options set the width of the next polyline segments you draw. Widths greater than zero produce wide lines, which are filled if Fill mode is on and outlined if Fill mode is off.

Intersections of adjacent wide segments are usually beveled. However, nontangent arc segments, acute angles, or segments that use a dash-dot linetype are not beveled.

Create Polylines from the Boundaries of Objects

You can create a polyline from the boundaries of objects that form a closed area with BOUNDARY. A polyline created using this method is a separate object, distinct from the objects used to create it.

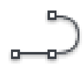
To expedite the boundary selection process in large or complex drawings, you can specify a group of boundary candidates, called a boundary set. You create this set by selecting the objects you want to use define the boundary.



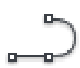
See also:

- Draw Rectangles and Polygons on page 807
- Modify Complex Objects on page 1153
- Break and Join Objects on page 1138
- Control Lineweights on page 614

To draw a polyline with straight segments

1. Click Home tab ➤ Draw panel ➤ Polyline. 
2. Specify the first point of the polyline.
3. Specify the endpoint of the first polyline segment.
4. Continue specifying segment endpoints as needed.
5. Press ENTER to end, or enter **c** to close the polyline.

   To start a new polyline at the endpoint of the last polyline drawn, start the PLINE command again and press ENTER at the Specify Start Point prompt.

Toolbar: Draw 
Command entry: PLINE

To draw a line and arc combination polyline

1. Click Home tab ➤ Draw panel ➤ Polyline. 
2. Specify the start point of the polyline segment.

3 Specify the endpoint of the polyline segment.
   - Switch to Arc mode by entering **a** (Arc) at the command prompt.
   - Return to Line mode by entering **L** (Line).
4 Specify additional polyline segments as needed.
5 Press ENTER to end, or enter **c** to close the polyline.

Toolbar: Draw
Command entry: PLINE

To create a wide polyline

1 Click Home tab ➤ Draw panel ➤ Polyline.
2 Specify the start point of the line segment.
3 Enter **w** (Width).
4 Enter the starting width of the line segment.
5 Specify the ending width of the line segment using one of the following methods:
   - To create a line segment of equal width, press ENTER.
   - To create a tapering line segment, enter a different width.
6 Specify the endpoint of the polyline segment.
7 Continue specifying segment endpoints as needed.
8 Press ENTER to end, or enter **c** to close the polyline.

Toolbar: Draw
Command entry: PLINE

To create a boundary polyline

1. Click Home tab ➤ Draw panel ➤ Boundary.
2. In the Boundary Creation dialog box, in the Object Type list, select Polyline.
3. Under Boundary Set, do one of the following:
   - To create a boundary set from all objects visible in the current viewport, select Current Viewport. Avoid this option for large, complex drawings.
   - To specify which objects to include in the boundary set, click New. Select the objects that you want to use to create the boundary. Using this option automatically selects the Existing Set option.
4. Click Pick Points.
5. Specify points within each area that you want to form a boundary polyline.

   This area must be totally enclosed; that is, there can be no gaps between enclosing objects. You can select more than one area. Click Island Detection if you want internal closed areas to be included in the boundary set.
6. Press ENTER to create the boundary polyline and end the command.

   The command creates a polyline in the shape of the boundary. Because this polyline overlaps the objects used to create it, it may not be visible. However, you can move, copy, or modify it just as you can any other polyline.

Command entry: BOUNDARY

Quick Reference

Commands

3DPOLY

Creates a 3D polyline.

BOUNDARY

Creates a region or a polyline from an enclosed area.

EXPLODE

Breaks a compound object into its component objects.

FILL

Controls the filling of objects such as hatches, 2D solids, and wide polylines.

PEDIT

Edits polylines and 3D polygon meshes.

PLINE

Creates a 2D polyline.

POLYGON

Creates an equilateral closed polyline.

RECTANG

Creates a rectangular polyline.

System Variables

FILLMODE

Specifies whether hatches and fills, 2D solids, and wide polylines are filled in.

HPBOUND

Controls the object type created by the BHATCH and BOUNDARY commands.

PLINECONVERTMODE

Specifies the fit method used in converting splines to polylines.

PLINEGEN

Sets how linetype patterns generate around the vertices of a 2D polyline.

PLINETYPE

Specifies whether optimized 2D polylines are used.

PLINEWID

Stores the default polyline width.

Utilities

No entries

Command Modifiers

No entries

Draw Rectangles and Polygons

You can create rectangles and regular polygons quickly. Creating polygons is a simple way to draw equilateral triangles, squares, pentagons, hexagons, and so on.

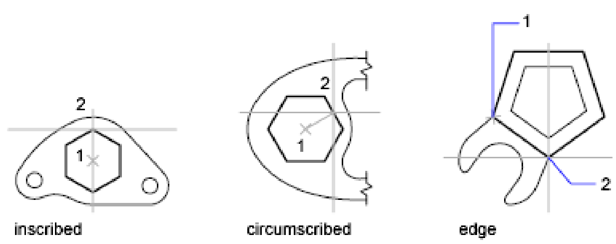
If necessary, you can use EXPLODE to convert the resulting polyline object into lines.

Draw Rectangles

Use RECTANG to create closed polylines in a rectangular shape.

Draw Regular Polygons

Use POLYGON to create closed polylines with between 3 and 1,024 equal-length sides. The following illustrations show polygons created using three methods. In each case, two points are specified.



See also:

- Draw Polylines on page 801

To draw a circumscribed polygon

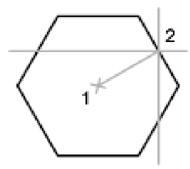
1 Click Home tab ➤ Draw panel ➤ Polygon. 

2 At the command prompt, enter the number of sides.

3 Specify the center of the polygon (1).

4 Enter **c** to specify a polygon circumscribed about a circle.

5 Enter the radius length (2).



Toolbar: Draw

Command entry: POLYGON

To draw a polygon by specifying one edge

1 Click Home tab ➤ Draw panel ➤ Polygon.

2 At the command prompt, enter the number of sides.

3 Enter **e** (Edge).

4 Specify the start point for one polygon segment.

5 Specify the endpoint of the polygon segment.

Toolbar: Draw

Command entry: POLYGON

To draw an inscribed polygon

1 Click Home tab ➤ Draw panel ➤ Polygon.

2 At the command prompt, enter the number of sides.

3 Specify the center of the polygon.

4 Enter **i** to specify a polygon inscribed within a circle of specified points.

5 Enter the radius length.

Toolbar: Draw
Command entry: POLYGON

To draw a rectangle

1 Click Home tab ➤ Draw panel ➤ Rectangle.

2 Specify the first corner of the rectangle.

3 Specify the other corner of the rectangle.

Toolbar: Draw
Command entry: RECTANG

Quick Reference

Commands

BOUNDARY

Creates a region or a polyline from an enclosed area.

EXPLODE

Breaks a compound object into its component objects.

POLYGON

Creates an equilateral closed polyline.

RECTANG

Creates a rectangular polyline.

System Variables

FILLMODE

Specifies whether hatches and fills, 2D solids, and wide polylines are filled in.

HPBOUND

Controls the object type created by the BHATCH and BOUNDARY commands.

PLINEWID

Stores the default polyline width.

POLYSIDES

Sets the default number of sides for the POLYGON command.

SNAPANG

Sets the snap and grid rotation angle for the current viewport relative to the current UCS.

Utilities

No entries

Command Modifiers

No entries

Draw Multiline Objects

Multilines are composed of 1 to 16 parallel lines, called elements.

When you draw a multiline, you can use the STANDARD style, which has two elements, or specify a style that you created previously. You can also change the justification and scale of the multiline before you draw it.

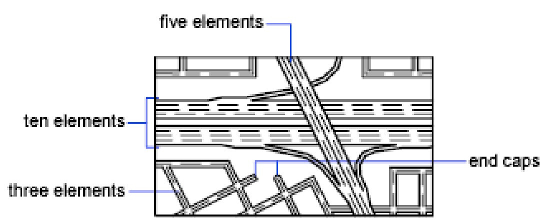
Multiline justification determines which side of the cursor that the multiline is drawn, or whether it is centered on the cursor.

Multiline scale controls the overall width of the multiline using the current units. Multiline scale does not affect linetype scale. If you change the multiline scale, you might need to make equivalent changes to the linetype scale to prevent dots or dashes from being disproportionately sized.

Create Multiline Styles

You can create named styles for multilines to control the number of elements and the properties of each element. The properties of multilines include

- The total number of elements and position of each element
- The offset distance for each element from the middle of the multiline
- The color and linetype of each element
- The visibility of the lines, called *joints*, that appear at each vertex
- The type of end caps that are used
- The background fill color of the multiline

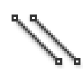


You can add up to 16 elements to a multiline style. Elements with a positive offset appear on one side of the middle of the multiline; elements with a negative offset appear on the other side of the middle of the multiline.

See also:

- Modify Multilines on page 1165

To draw a multiline

1 Click Draw menu ➤ Multiline. 

2 At the command prompt, enter **st** to select a style.

3 To list available styles, enter the style name or enter **?**.

4 To justify the multiline, enter **j** and select top, zero, or bottom justification.

5 To change the scale of the multiline, enter **s** and enter a new scale.

Now draw the multiline.

6 Specify the starting point.

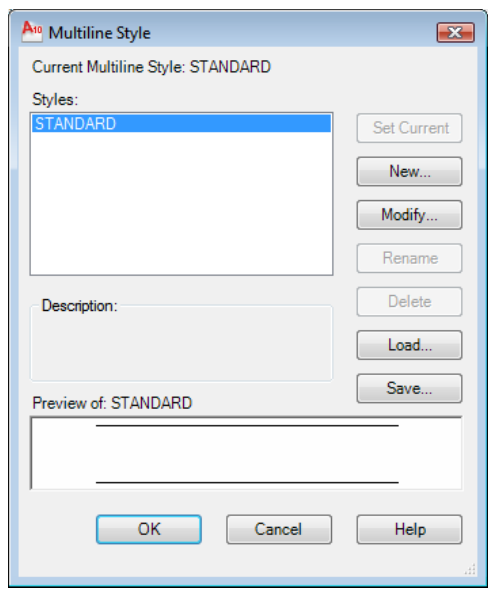
7 Specify a second point.

8 Specify additional points, or press ENTER. If you specify three or more points, you can enter **c** to close the multiline.

Command entry: MLINE

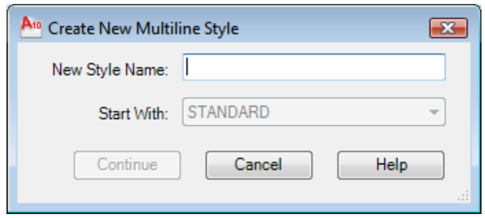
To create a multiline style

1 Click Format menu ➤ Multiline Style. 



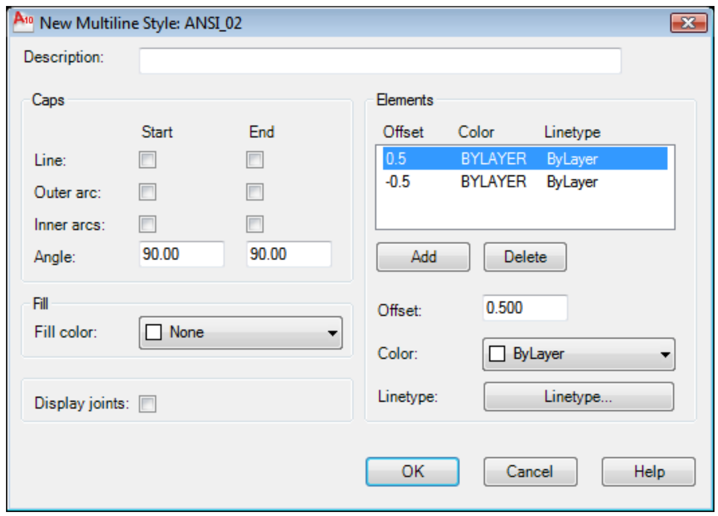
2 In the Multiline Style dialog box, click New.

3 In the Create New Multiline Style dialog box, enter a name for the multiline style and select a multiline style from which to start. Click Continue.



4 In the New Multiline Style dialog box, select the parameters for the multiline style. You can also enter a description.

Descriptions are optional and can be up to 255 characters, including spaces.



5 Click OK.

6 In the Multiline Style dialog box, click Save to save the multiline style to a file (the default is *acad.mln*). You can save multiline styles to the same file.

If you create more than one multiline style, save the current style before creating a new one or you lose the changes to the first style.

Quick Reference

Commands

OFFSET

Creates concentric circles, parallel lines, and parallel curves.

MLINE

Creates multiple parallel lines.

MLSTYLE

Creates, modifies, and manages multiline styles.

System Variables

CMLSTYLE

Sets the multiline style that governs the appearance of the multiline.

Utilities

No entries

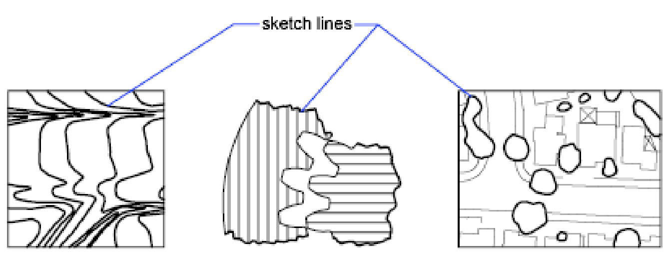
Command Modifiers

No entries

Draw Freehand Sketches

Sketching is useful for creating irregular boundaries or for tracing with a digitizer.

You can use the SKETCH command to draw freehand sketches. Sketching is useful for creating irregular boundaries or for tracing with a digitizer.



Create Sketches

To sketch, use the pointing device like a pen, clicking to put the “pen” down on the screen to draw and clicking again to lift it up and stop drawing. Freehand sketches comprise many line segments. Each line segment can be a separate object or a polyline. You set the minimum length or increment of the segments. Small line segments allow for greater accuracy, but they can greatly increase the drawing file size. For this reason, use this tool sparingly.

Before sketching, check the CELTYPE system variable to make sure the current linetype is BYLAYER. If you use a linetype with dots or dashes and set the sketch line segment shorter than the spaces or dashes, you won't see the spaces or dashes.

Erase Freehand Lines

You erase freehand lines by using the Erase option of the SKETCH command. In Erase mode, wherever the cursor intersects the freehand line, everything from the intersection to the end of the line is erased.

Once you record freehand lines, you can't edit them or erase them with the Erase option of SKETCH. Use the ERASE command after you finish sketching.

Sketch in Tablet Mode

You use Tablet mode with a digitizer. Sketching in Tablet mode is useful for such things as tracing map outlines from paper directly into a drawing. You can't turn off Tablet mode while sketching.

When Tablet mode is on, you can configure the program to map the paper drawing's coordinate system directly into the world coordinate system. Thus, there is a direct correlation between the coordinates where screen crosshairs appear, the coordinates on the tablet, and the coordinates in the original paper drawing. After configuring the program to match the coordinates of the paper drawing, you may find that the area shown on the screen is not the area you need. To avoid this problem, use ZOOM to display the entire work area before you start to sketch.

With some digitizers you can't select the menus while Tablet mode is on. See your digitizer documentation for details.

Maintain Sketching Accuracy

To ensure accuracy on a slow computer, set the record increment value to a negative value. SKETCH uses this value as if it was positive but tests every point received from the pointer against twice the record increment. If the point is more than two record increments away, your computer beeps as a warning that you should slow down to avoid losing accuracy. For example, if the record increment is -1, you should move the cursor in increments of no more than 2. Using this method does not slow down the tracing speed.

To sketch and record freehand lines

1 At the command prompt, enter **sketch**.

2 At the Record Increment prompt, enter the minimum line segment length.

3 Click the start point to put the “pen” down.

   When you move the pointing device, temporary freehand line segments of the length you specified are drawn. SKETCH doesn't accept coordinate input. During the command, freehand lines are displayed in a different color.

4 Click the endpoint to lift the “pen” up so that you can move the cursor around the screen without drawing. Click a new start point to resume drawing from the new cursor position.

5 Enter **r** at any time to record (save) the line you're drawing in the database and those already drawn.

   If the pen is down, you can continue drawing after recording. If the pen is up, click to resume drawing. The freehand line starts from wherever the cursor is when you click.

6 Press ENTER to complete the sketch and record all unrecorded lines.

To erase freehand lines

1 While running the SKETCH command, with the pen up or down, enter **e** (Erase).

   If the pen was down, it moves up.

2 Move the cursor to the end of the line you drew last and then move it back as far along the line as you want to erase.

3 To end the erasure and return to the SKETCH Command prompt, enter **p**. To undo the erasure, enter **e**.

   If you want to change the current viewport while sketching, make sure the pen is up, all lines entered so far have been recorded, and Tablet mode is off.

Quick Reference

Commands

SKETCH

Creates a series of freehand line segments.

System Variables

SKETCHINC

Sets the record increment for the SKETCH command.

SKPOLY

Determines whether the SKETCH command generates lines or polylines.

Utilities

No entries

Command Modifiers

No entries

Draw Curved Objects

Curved objects are arcs, circles, polyline arcs, donuts, ellipses, and splines.

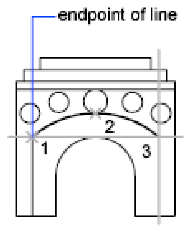
Draw Arcs

To create an arc, you can specify various combinations of center, endpoint, start point, radius, angle, chord length, and direction values.

You can create arcs in several ways. With the exception of the first method, arcs are drawn counterclockwise from the start point to the endpoint.

Draw Arcs by Specifying Three Points

You can create an arc by specifying three points. In the following example, the start point of the arc snaps to the endpoint of a line. The second point of the arc snaps to the middle circle in the illustration.

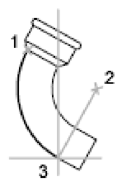


Draw Arcs by Specifying Start, Center, End

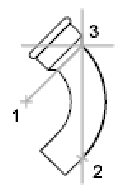
You can create an arc using a start point, center, and a third point that determines the endpoint.

The distance between the start point and the center determines the radius. The endpoint is determined by a line from the center that passes through the third point. The resulting arc is always created counterclockwise from the start point.

Using different options, you can specify either the start point first or the center point first.



start(1), center(2), end(3)



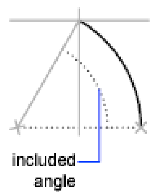
center(1), start(2), end(3)

Draw Arcs by Specifying Start, Center, Angle

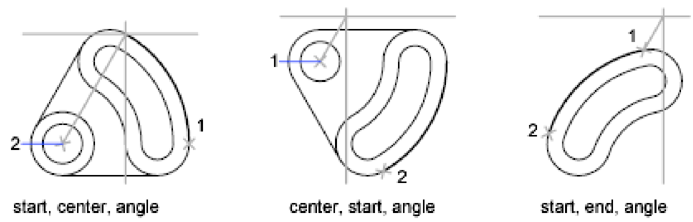
You can create an arc using a start point, center, and an included angle.

The distance between the start point and the center determines the radius. The other end of the arc is determined by specifying an included angle that uses the center of the arc as the vertex. The resulting arc is always created counterclockwise from the start point.

Using different options, you can specify either the start point first or the center point first.



The included angle determines the endpoint of the arc. Use the Start, End, Angle method when you know both endpoints but cannot snap to a center point.

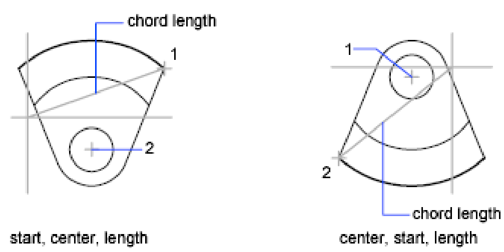


Draw Arcs by Specifying Start, Center, Length

You can create an arc using a start point, center, and the length of a chord.

The distance between the start point and the center determines the radius. The other end of the arc is determined by specifying the length of a chord between the start point and the endpoint of the arc. The resulting arc is always created counterclockwise from the start point.

Using different options, you can specify either the start point first or the center point first.



The length of the chord of the arc determines the included angle.

Draw Arcs by Specifying Start, End, Angle

You can create an arc using a start point, endpoint, and an included angle.

The included angle between the endpoints of the arc determines the center and the radius of the arc.

Draw Arcs by Specifying Start, End, Direction

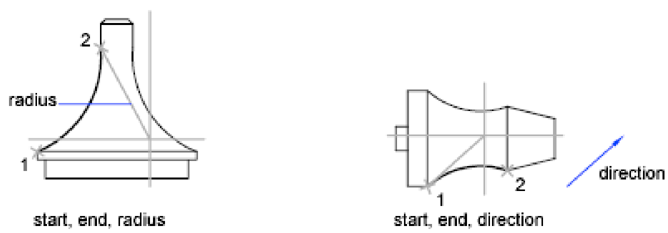
You can create an arc using a start point, endpoint, and a tangent direction at the start point.

The tangent direction can be specified either by locating a point on the desired tangent line, or by entering an angle. You can determine which endpoint controls the tangent by changing the order in which you specify the two endpoints.

Draw Arcs by Specifying Start, End, Radius

You can create an arc using a start point, endpoint, and a radius.

The direction of the bulge of the arc is determined by the order in which you specify its endpoints. You can specify the radius either by entering it or by specifying a point at the desired radius distance.



Draw Contiguous Tangent Arcs and Lines

Immediately after you create an arc, you can start a line that is tangent to the arc at an endpoint by starting the LINE command and pressing ENTER at the Specify First Point prompt. You need to specify only the line length.

Immediately after you create a line or an arc, you can start an arc that is tangent at an endpoint by starting the ARC command and pressing ENTER at the Specify Start Point prompt. You need to specify only the endpoint of the new arc.

See also:

- Draw Polylines on page 801
- Break and Join Objects on page 1138

To draw an arc by specifying three points

1 Click Home tab ➤ Draw panel ➤ 3-Point.

2 Specify the start point.

3 Specify a point on the arc.

4 Specify the endpoint.

Toolbar: Draw
Command entry: ARC

To draw an arc using a start point, a center point, and an endpoint

1 Click Home tab ➤ Draw panel ➤ Start, Center, End.

2 Specify a start point.

3 Specify the center point.

4 Specify the endpoint.

Toolbar: Draw
Command entry: ARC

To continue an arc with a tangential line

1 Complete the arc.

2 Click Home tab ➤ Draw panel ➤ Line.

3 Press ENTER at the first prompt.

4 Enter the length of the line and press ENTER.

Toolbar: Draw
Command entry: LINE

To continue an arc with a tangential arc

1 Complete the arc.

2 Click Home tab ➤ Draw panel ➤ Continue.

3 Specify the second endpoint of the tangent arc.

Toolbar: Draw
Command entry: ARC

Quick Reference

Commands

ARC

Creates an arc.

LINE

Creates straight line segments.

OFFSET

Creates concentric circles, parallel lines, and parallel curves.

VIEWRES

Sets the resolution for objects in the current viewport.

System Variables

ANGDIR

Sets the direction of positive angles.

LASTANGLE

Stores the end angle of the last arc entered relative to the XY plane of the current UCS for the current space.

WHIPARC

Controls whether the display of circles and arcs is smooth.

Utilities

No entries

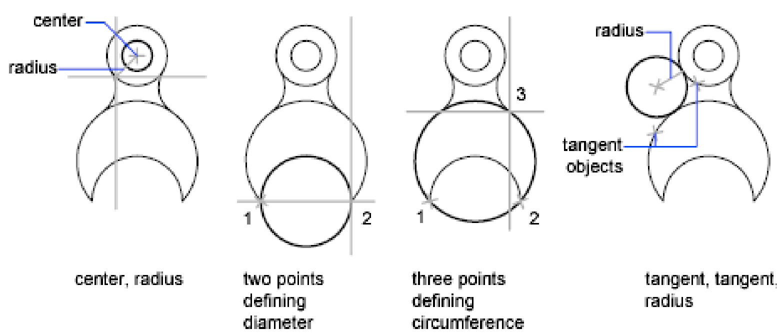
Command Modifiers

No entries

Draw Circles

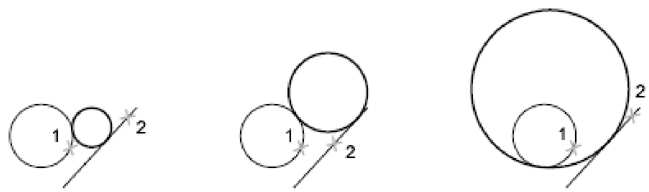
To create circles, you can specify various combinations of center, radius, diameter, points on the circumference, and points on other objects.

You can create circles in several ways. The default method is to specify the center and the radius. Three other ways to draw a circle are shown in the illustration.



Draw a Circle Tangent to Other Objects

The tangent point is a point where an object touches another object without intersecting it. To create a circle that is tangent to other objects, select the objects and then specify the radius of the circle. In the illustrations below, the bold circle is the one being drawn, and points 1 and 2 select the objects to which it is tangent.



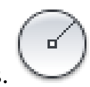
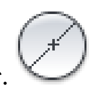
radius of new circle = 1 | radius of new circle = 2 | radius of new circle = 4

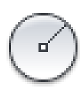
To create a circle tangent at three points, set running object snaps (OSNAP) to Tangent and use the three-point method to create the circle.

See also:

- Use Object Snaps on page 676
- Draw Isometric Circles

To draw a circle by specifying a center point and radius or diameter

1. Do one of the following:
   - Click Home tab ➤ Draw panel ➤ Center, Radius. 
   - Click Home tab ➤ Draw panel ➤ Center, Diameter. 
2. Specify the center point.
3. Specify the radius or diameter.

Toolbar: Draw 
Command entry: CIRCLE

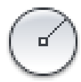
To create a circle tangent to two objects

1. Click Home tab ➤ Draw panel ➤ Tan, Tan, Radius. 
   The command starts Tangent object snap mode.

2 Select the first object to draw the circle tangent to.

3 Select the second object to draw the circle tangent to.

4 Specify the radius of the circle.

Toolbar: Draw 
Command entry: CIRCLE

Quick Reference

Commands

CIRCLE

Creates a circle.

OFFSET

Creates concentric circles, parallel lines, and parallel curves.

System Variables

CIRCLERAD

Sets the default circle radius.

WHIPARC

Controls whether the display of circles and arcs is smooth.

Utilities

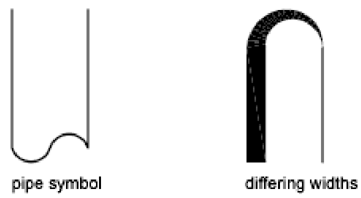
No entries

Command Modifiers

No entries

Draw Polyline Arcs

A polyline is a connected sequence of line segments created as a single object. You can create straight line segments, arc segments, or a combination of the two.



Multisegmented lines provide editing capabilities unavailable for single lines. For example, you can adjust their width and curvature. After you've created a polyline, you can edit it with PEDIT or use EXPLODE to convert it to individual line and arc segments. You can

- Convert a spline-fit polyline into a true spline with SPLINE
- Use closed polylines to create a polygon
- Create a polyline from the boundaries of overlapping objects

Create Arc Polylines

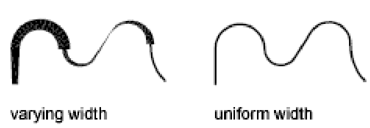
When you draw arc segments in a polyline, the first point of the arc is the endpoint of the previous segment. You can specify the angle, center point, direction, or radius of the arc. You can also complete the arc by specifying a second point and an endpoint.

Create Closed Polylines

You can draw a closed polyline to create a polygon. To close a polyline, specify the starting point of the last side of the object, enter c (Close), and press ENTER.

Create Wide Polylines

You can draw polylines of various widths by using the Width and Halfwidth options. You can set the width of individual segments and make them taper gradually from one width to another. These options become available after you specify a starting point for the polyline.



The Width and Halfwidth options set the width of the next polyline segments you draw. Zero (0) width produces a thin line. Widths greater than zero produce wide lines, which are filled if Fill mode is on and outlined if Fill mode is off. The Halfwidth option sets width by specifying the distance from the center of the wide polyline to an outside edge.

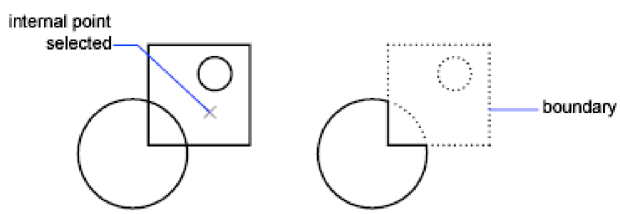
Taper

When you use the Width option, you are prompted for both a starting and an ending width. By entering different values, you can taper the polyline. The starting and ending points of wide polyline segments are in the center of the line. Intersections of adjacent wide segments are usually beveled. However, nontangent arc segments, acute angles, or segments that use a dash-dot linetype are not beveled.

Create Polylines from the Boundaries of Objects

You can create a polyline from the boundaries of overlapping objects that form a closed area. A polyline created using the boundary method is a separate object, distinct from the objects used to create it. You can edit it using the same methods used to edit other polylines.

To expedite the boundary selection process in large or complex drawings, you can specify a group of boundary candidates, called a boundary set. You create this set by selecting the objects you want to use to define the boundary.



See also:

- Modify Splines on page 1160
- Modify or Join Polylines on page 1155
- Break and Join Objects on page 1138
- Control Lineweights on page 614

To draw a polyline with straight segments

1 Click Home tab ➤ Draw panel ➤ Polyline.

2 Specify the first point of the polyline.

3 Specify the endpoint of the first polyline segment.

4 Continue specifying segment endpoints as needed.

5 Press ENTER to end, or enter **c** to close the polyline.

   To start a new polyline at the endpoint of the last polyline drawn, start the PLINE command again and press ENTER at the Specify Start Point prompt.

Toolbar: Draw
Command entry: PLINE

To draw a line and arc combination polyline

1 Click Home tab ➤ Draw panel ➤ Polyline.

2 Specify the start point of the polyline segment.

3 Specify the endpoint of the polyline segment.

   - Switch to Arc mode by entering **a** (Arc) at the command prompt.
   - Return to Line mode by entering **L** (Line).

4 Specify additional polyline segments as needed.

5 Press ENTER to end, or enter **c** to close the polyline.

Toolbar: Draw
Command entry: PLINE

To create a wide polyline

1 Click Home tab ➤ Draw panel ➤ Polyline.

2 Specify the start point of the line segment.

3 Enter **w** (Width).

4 Enter the starting width of the line segment.

5 Specify the ending width of the line segment using one of the following methods:

- To create a line segment of equal width, press ENTER.
- To create a tapering line segment, enter a different width.

6 Specify the endpoint of the polyline segment.

7 Continue specifying segment endpoints as needed.

8 Press ENTER to end, or enter **c** to close the polyline.

Toolbar: Draw
Command entry: PLINE

To create a boundary polyline

1 Click Home tab ➤ Draw panel ➤ Boundary.

2 In the Boundary Creation dialog box, in the Object Type list, select Polyline.

3 Under Boundary Set, do one of the following:

- To create a boundary set from all objects visible in the current viewport, select Current Viewport in the list. Avoid this option for large, complex drawings.
- To specify which objects to include in the boundary set, click New. Select the objects that you want to use to create the boundary. Choosing this option automatically selects the Existing Set option.

4 Click Pick Points.

5 Specify points within each area that you want to form a boundary polyline.

   This area must be totally enclosed; that is, there can be no gaps between enclosing objects. You can select more than one area.

6 Press ENTER to create the boundary polyline and end the command.

   The command creates a polyline in the shape of the boundary. Because this polyline overlaps the objects used to create it, it may not be visible. However, you can move, copy, or modify it just as you can any other polyline.

Command entry: BOUNDARY

Quick Reference

Commands

3DPOLY

Creates a 3D polyline.

BOUNDARY

Creates a region or a polyline from an enclosed area.

OFFSET

Creates concentric circles, parallel lines, and parallel curves.

PEDIT

Edits polylines and 3D polygon meshes.

PLINE

Creates a 2D polyline.

VIEWRES

Sets the resolution for objects in the current viewport.

System Variables

FILLMODE

Specifies whether hatches and fills, 2D solids, and wide polylines are filled in.

HPBOUND

Controls the object type created by the BHATCH and BOUNDARY commands.

PLINECONVERTMODE

Specifies the fit method used in converting splines to polylines.

PLINEGEN

Sets how linetype patterns generate around the vertices of a 2D polyline.

PLINETYPE

Specifies whether optimized 2D polylines are used.

PLINEWID

Stores the default polyline width.

Utilities

No entries

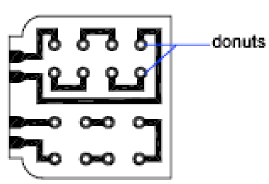
Command Modifiers

No entries

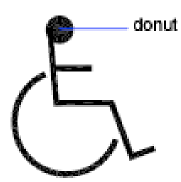
Draw Donuts

Donuts are filled rings or solid-filled circles that actually are closed polylines with width.

To create a donut, you specify its inside and outside diameters and its center. You can continue creating multiple copies with the same diameter by specifying different center points. To create solid-filled circles, specify an inside diameter of 0.



filled rings



solid-filled circles

To create a donut

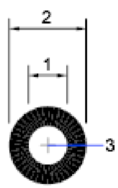
1 Click Home tab ➤ Draw panel ➤ Donut. 

2 Specify the inside diameter (1).

3 Specify the outside diameter (2).

4 Specify the center of the donut (3).

5 Specify the center point for another donut, or press ENTER to complete the command.



Quick Reference

Commands

DONUT

Creates a filled circle or a wide ring.

FILL

Controls the filling of objects such as hatches, 2D solids, and wide polylines.

System Variables

DONUTID

Sets the default for the inside diameter of a donut.

DONUTOD

Sets the default for the outside diameter of a donut.

FILLMODE

Specifies whether hatches and fills, 2D solids, and wide polylines are filled in.

Utilities

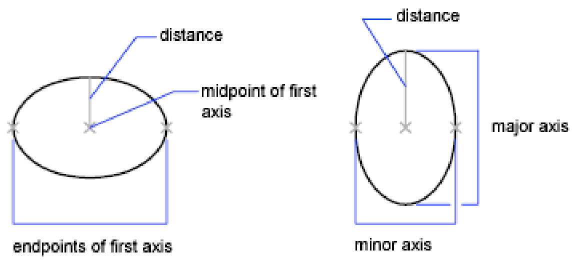
No entries

Command Modifiers

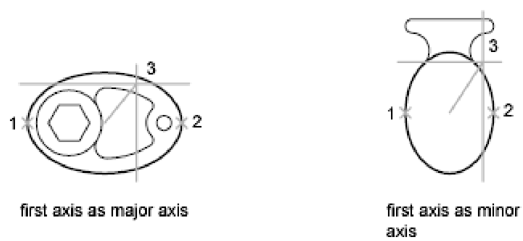
No entries

Draw Ellipses

The shape of an ellipse is determined by two axes that define its length and width. The longer axis is called the major axis, and the shorter one is the minor axis.



The illustrations below show two different ellipses created by specifying axis and distance. The third point specifies only a distance and does not necessarily designate the axis endpoint.



If you are drawing on isometric planes to simulate 3D, you can use ellipses to represent isometric circles viewed from an oblique angle. First you need to turn on Isometric Snap in the Drafting Settings dialog box.

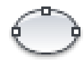
See also:

- Draw Isometric Circles
- Break and Join Objects on page 1138

To draw an isometric circle

1 Click Tools menu ➤ Drafting Settings.

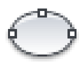
2 In the Drafting Settings dialog box, Snap and Grid tab, under Snap Type and Style, click Isometric Snap. Click OK.

3 Click Home tab ➤ Draw panel ➤ Axis, End. 

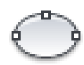
4 Enter **i** (Isocircle).

5 Specify the center of the circle.

6 Specify the radius or diameter of the circle.

Toolbar: Draw 
Command entry: ELLIPSE

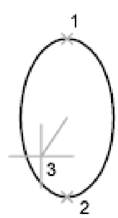
To draw a true ellipse using endpoints and distance

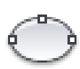
1 Click Home tab ➤ Draw panel ➤ Axis, End. 

2 Specify the first endpoint of the first axis (1).

3 Specify the second endpoint of the first axis (2).

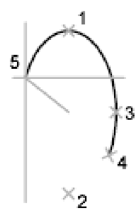
4 Drag the pointing device away from the midpoint, and click to specify a distance (3) for half the length of the second axis.

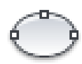


Toolbar: Draw 
Command entry: ELLIPSE

To draw an elliptical arc using start and end angles

1. Click Home tab ➤ Draw panel ➤ Arc. 
2. Specify endpoints for the first axis (1 and 2).
3. Specify a distance to define half the length of the second axis (3).
4. Specify the start angle (4).
5. Specify the end angle (5).
   The elliptical arc is drawn counterclockwise between the start point and endpoint.



Toolbar: Draw 
Command entry: ELLIPSE

Quick Reference

Commands

ELLIPSE

Creates an ellipse or an elliptical arc.

System Variables

ANGDIR

Sets the direction of positive angles.

PELLIPSE

Controls the ellipse type created with ELLIPSE.

Utilities

No entries

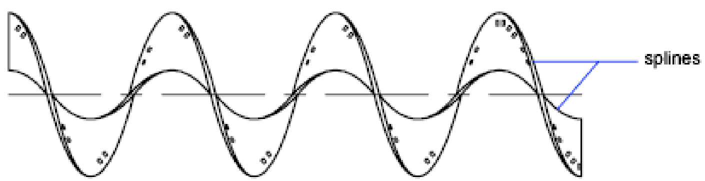
Command Modifiers

No entries

Draw Splines

A spline is a smooth curve that passes through or near a given set of points. You can control how closely the curve fits the points.

The SPLINE command creates a particular type of spline known as a nonuniform rational B-spline (NURBS) curve. A NURBS curve produces a smooth curve between control points.



You create splines by specifying points. You can close the spline so that the start and endpoints are coincident and tangent.

Tolerance describes how closely the spline fits the set of fit points you specify. The lower the tolerance, the more closely the spline fits the points. At zero tolerance, the spline passes through the points. You can change the spline-fitting tolerance while drawing the spline to see the effect.

You can use two methods for creating splines:

- Create spline curves with the Spline option of PEDIT to smooth existing polylines created with PLINE. Such spline-fit polylines are created with uniform knot vectors and are more likely to be included in drawings created with earlier versions of the product.
- Create splines, which are NURBS curves, with SPLINE. Drawings containing splines use less memory and disk space than those containing spline-fit polylines of similar shape.

You can easily convert spline-fit polylines into true splines with SPLINE.

See also:

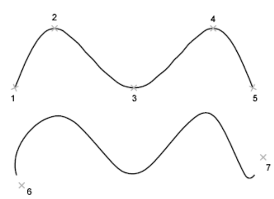
- [Modify Splines](#) on page 1160
- [Break and Join Objects](#) on page 1138

To convert a spline-fit polyline to a spline

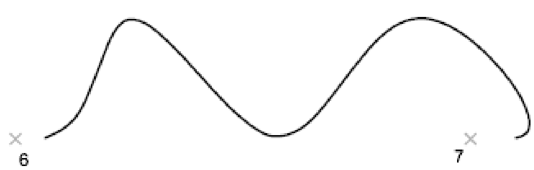
1 Click Home tab ➤ Draw panel ➤ Spline. 
2 Enter **o** (Object).
3 Select a spline-fit polyline and press ENTER.
The selected object changes from a polyline to a spline.

To convert a spline by specifying points

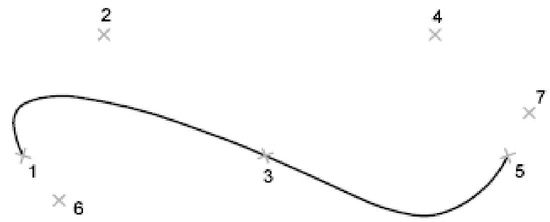
1 Click Home tab ➤ Draw panel ➤ Spline. 
2 Specify the start point for the spline (1).
3 Specify points (2 through 5) to create the spline, and press ENTER.
4 Specify the start and end tangents (6, 7).



The spline below uses the same points but different start and end tangents.



The spline below uses the same points but a higher tolerance and different start and end tangents.



Toolbar: Draw

Command entry: SPLINE

Quick Reference

Commands

PEDIT

Edits polylines and 3D polygon meshes.

PLINE

Creates a 2D polyline.

SPLINE

Creates a smooth curve that passes through or near specified points.

SPLINEDIT

Edits a spline or spline-fit polyline.

System Variables

PLINECONVERTMODE

Specifies the fit method used in converting splines to polylines.

Utilities

No entries

Command Modifiers

No entries

Draw Helixes

A helix is an open 2D or 3D spiral.

You can use a helix as a path with the SWEEP command. For example, you might sweep a circle along a helix path to create a solid model of a spring.

When you create a helix, you can specify the following:

- Base radius
- Top radius
- Height
- Number of turns
- Turn height
- Twist direction

If you specify the same value for both the base radius and the top radius, then a cylindrical helix is created. By default, the top radius is set to the same value as the base radius. You cannot specify 0 for both the base radius and top radius.

If you specify different values for the top radius and the base radius, then a conical helix is created.

If you specify a height value of 0, then a flat, 2D spiral is created.

NOTE A helix is a spline approximation of a real helix. Length values may not be completely accurate. However, when you use a helix as a sweep path, the resulting values will be accurate regardless of the approximation.

See also:

- Modify Helixes on page 1163
- Create a Solid or Surface by Sweeping on page 1202

To create a helix

1 Click Home tab ➤ Draw panel ➤ Helix.

2 Specify the center point for the base of the helix.

3 Specify the base radius.

4 Specify the top radius or press ENTER to specify the same value as the base radius.

5 Specify the height of the helix.

Toolbar: Modeling
Command entry: HELIX

Quick Reference

Commands

HELIX

Creates a 2D spiral or 3D spring.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Draw Construction and Reference Geometry

Construction lines and reference points are temporary objects you create to help you draw accurately.

Draw Reference Points

Point objects are useful as nodes or reference geometry for object snaps and relative offsets.

You can set the style of the points and their size relative to the screen or in absolute units. Changing the style of points

- Makes them more visible and easier to differentiate from grid dots
- Affects the display of all point objects in the drawing
- Requires using REGEN to make the change visible

To set point style and size

1. Click Format menu ➤ Point Style.
2. In the Point Style dialog box, select a point style.
3. In the Point Size box, specify a size, either relative to the screen or in absolute units.
4. Click OK.

Command entry: DDPTYPE

To create a point object

1. Click Home tab ➤ Draw panel ➤ Point drop-down ➤ Multiple Points.
2. Specify the point location.

   You can snap to a point using the Node object snap.

Toolbar: Draw
Command entry: POINT

Quick Reference

Commands

DDPTYPE

Specifies the display style and size of point objects.

POINT

Creates a point object.

System Variables

PDMODE

Controls how point objects are displayed.

PDSIZE

Sets the display size for point objects.

Utilities

No entries

Command Modifiers

No entries

Draw Construction Lines (and Rays)

Lines that extend to infinity in one or both directions, known as rays and construction lines, respectively, can be used as references for creating other objects.

For example, you can use construction lines to find the center of a triangle, prepare multiple views of the same item, or create temporary intersections to use for object snaps.

Infinite lines do not change the total area of the drawing. Therefore, their infinite dimensions have no effect on zooming or viewpoints, and they are ignored by commands that display the drawing extents. You can move, rotate, and copy infinite lines just as you can move, rotate, and copy other objects. You may want to create infinite lines on a construction line layer that can be frozen or turned off before plotting.

Construction Lines

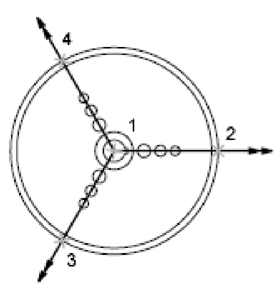
A construction line () can be placed anywhere in three-dimensional space. You can specify its orientation in several ways. The default method for creating the line is the two-point method: you specify two points to define the orientation. The first point, the root, is the conceptual midpoint of the construction line, that is, the point snapped to by the Midpoint object snap.

You can also create construction lines in several other ways.

- **Horizontal and Vertical**. Create construction lines that pass through a point you specify and are parallel to the *X* or *Y* axis of the current UCS.
- **Angle**. Creates a construction line in one of two ways. Either you select a reference line and then specify the angle of the construction line from that line, or you create a construction line at a specific angle to the horizontal axis by specifying an angle and then a point through which the construction line should pass.
- **Bisector**. Creates a construction line that bisects an angle you specify. You specify the vertex and the lines that create the angle.
- **Offset**. Creates a construction line parallel to a baseline you specify. You specify the offset distance, select the baseline, and then indicate on which side of the baseline to locate the construction line.

Rays

A ray is a line in three-dimensional space that starts at a point you specify and extends to infinity. Unlike construction lines, which extend in two directions, rays extend in only one direction. Using rays instead of construction lines can help reduce visual clutter. Like construction lines, rays are ignored by commands that display the drawing extents.



three rays

To create a construction line by specifying two points

1. Click Home tab ➤ Draw panel ➤ Construction Line.
2. Specify a point to define the root of the construction line.
3. Specify a second point through which the construction line should pass.
4. Continue to specify construction lines as needed.
   All subsequent xlines pass through the first point specified.
5. Press ENTER to end the command.

Toolbar: Draw
Command entry: XLINE

To create a ray

1. Click Home tab ➤ Draw panel ➤ Ray.
2. Specify a starting point for the ray.
3. Specify a point through which the ray should pass.
4. Continue to specify points to create additional rays as needed.
   All subsequent rays pass through the first point specified.
5. Press ENTER to end the command.

Command entry: RAY

Quick Reference

Commands

RAY

Creates a line that starts at a point and continues to infinity.

XLINE

Creates a line of infinite length.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

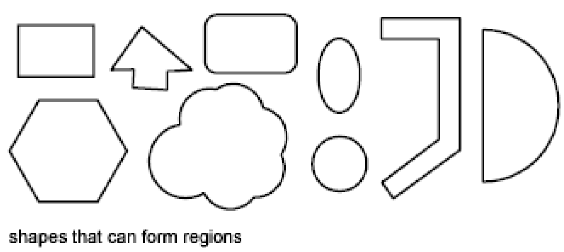
Create and Combine Areas (Regions)

Regions are two-dimensional enclosed areas that have physical properties such as centroids or centers of mass. You can combine existing regions into a single, complex region to calculate area.

Regions are two-dimensional enclosed areas you create from objects that form closed loops. Loops can be combinations of lines, polylines, circles, arcs, ellipses, elliptical arcs, and splines. The objects that make up the loops must either be closed or form closed areas by sharing endpoints with other objects.

Regions can be used for

- Applying hatching and shading
- Analyzing properties, such as area, using MASSPROP
- Extracting design information, such as the centroid



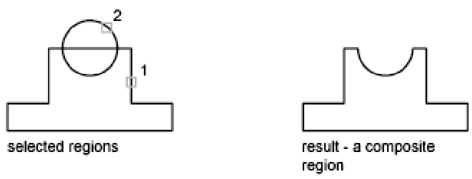
You can create regions out of multiple loops and out of open curves whose endpoints are connected and form loops. You cannot form regions from open objects that intersect to form a closed area: for example, intersecting arcs or self-intersecting curves.

You can also create regions using BOUNDARY.

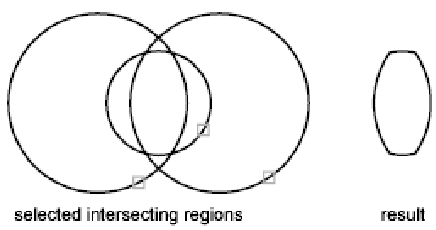
You create composite regions by combining, subtracting, or finding the intersection of regions. After forming these more complex regions, you can apply hatching or analyze their area.

Objects combined using UNION:

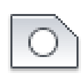
Objects combined using SUBTRACT:



Objects combined using INTERSECT:



To define regions

1 Click Home tab ➤ Draw panel ➤ Region. 

2 Select objects to create the region.

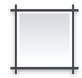
   These objects must each form an enclosed area, such as a circle or a closed polyline.

3 Press ENTER.

   A message at the command prompt indicates how many loops were detected and how many regions were created.

Command entry: REGION

To define regions by using boundaries

1 Click Home tab ➤ Draw panel ➤ Boundary. 

2 In the Boundary Creation dialog box, in the Object Type list, select Region.

3 Click Pick Points.

4 Specify a point in your drawing inside each closed area that you want to define as a region and press ENTER.

   This point is known as the internal point.

   NOTE You can make a new boundary set to limit the objects used to determine the boundary.

Command entry: BOUNDARY

To combine regions by adding

1 Click Home tab ➤ Solid Editing panel ➤ Union. 

2 Select one region for the union.

3 Select another region.

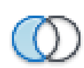
   You can select regions to unite in any order.

4 Continue selecting regions or press ENTER to end the command.

   The command converts the selected regions to a new combined region.

Command entry: UNION

To combine regions by subtracting

1 Click Home tab ➤ Solid Editing panel ➤ Subtract. 

2 Select one or more regions from which to subtract and press ENTER.

3 Select the region to subtract and press ENTER.

The areas of the second regions you selected are subtracted from the areas of the first regions.

Command entry: SUBTRACT

To combine regions by finding intersections

1 Click Home tab ➤ Solid Editing panel ➤ Intersect. 

2 Select one region of the intersection.

3 Select another intersecting region.

You can select regions in any order to find their intersection.

4 Continue selecting regions or press ENTER to end the command.

The command converts the selected regions to a new region defined by the intersection of the selected regions.

Command entry: INTERSECT

Quick Reference

Commands

BOUNDARY

Creates a region or a polyline from an enclosed area.

INTERSECT

Creates a 3D solid, surface, or 2D region from overlapping solids, surfaces, or regions.

MASSPROP

Calculates the mass properties of regions or 3D solids.

REGION

Converts an object that encloses an area into a region object.

SUBTRACT

Combines selected 3D solids, surfaces, or 2D regions by subtraction.

UNION

Combines selected 3D solids, surfaces, or 2D regions by addition.

System Variables

DELOBJ

Controls whether geometry used to create 3D objects is retained or deleted.

Utilities

No entries

Command Modifiers

No entries

Create Revision Clouds

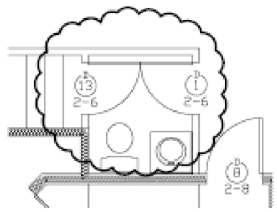
Revision clouds are polylines that consist of sequential arcs. They are used to call attention to parts of a drawing during the review stage.

If you review or redline drawings, you can increase your productivity by using the Revision Cloud feature to highlight your markups. REVCLOUD creates a polyline of sequential arcs to form a cloud-shaped object. You can select a style for a revision cloud: Normal or Calligraphy. If you select Calligraphy, the revision cloud looks as if it was drawn with a calligraphy pen.

You can create a revision cloud from scratch, or you can convert objects, such as a circle, ellipse, polyline, or spline, to a revision cloud. When you convert an object to a revision cloud, the original object is deleted if DELOBJ is set to 1 (the default).

You can set the minimum and maximum default values for the arc lengths of a revision cloud. When you draw a revision cloud, you can vary the size of the arcs by using pick points for the smaller arc segments. You can also edit the individual arc lengths and chord lengths of a revision cloud by adjusting the pick points.

REVCLOUD stores the last used arc length as a multiple of the DIMSCALE system variable to provide consistency among drawings with different scale factors.



Make sure that you can see the entire area to be outlined with REVCLOUD before you begin the command. REVCLOUD is not designed to support transparent and real-time panning and zooming.

To create a revision cloud from scratch

1 Click Home tab ➤ Draw panel ➤ Revision Cloud. 

2 At the command prompt, specify a new minimum and maximum arc length or specify a revision cloud starting point.

   The default minimum and maximum arc lengths are set to 0.5000 units. The maximum arc length can be no more than three times the minimum arc length.

3 Guide the crosshairs along the cloud path. You can click pick points along the path if you want to vary the size of the arcs.

4 Press ENTER at any time to stop drawing the revision cloud.

   To close the revision cloud, return to its starting point.

Toolbar: Draw 
Command entry: REVCLOUD

To create revision clouds with a calligraphy pen style

1 Click Home tab ➤ Draw panel ➤ Revision Cloud. 

2 At the command prompt, enter **style**.

3 At the command prompt, enter **calligraphy**.

4 Press ENTER to save the calligraphy setting and to continue with the command, or press ESC to end the command.

Toolbar: Draw
Command entry: REVCLOUD

To convert an object to a revision cloud

1 Click Home tab ➤ Draw panel ➤ Revision Cloud.

2 At the command prompt, specify a new minimum and maximum arc length or press ENTER.

   The default minimum and maximum arc lengths are set to 0.5000 units. The maximum arc length can be no more than three times the minimum arc length.

3 Select the circle, ellipse, polyline, or spline that you want to convert to a revision cloud.

   To reverse the direction of the arcs, enter **yes** at the command prompt and press ENTER.

4 Press ENTER to change the selected object to a revision cloud.

Toolbar: Draw
Command entry: REVCLOUD

To change the default values for arc lengths in a revision cloud

1 Click Home tab ➤ Draw panel ➤ Revision Cloud.

2 At the command prompt, specify a new minimum arc length and press ENTER.

3 At the command prompt, specify a new maximum arc length and press ENTER.

   The maximum arc length can be no more than three times the minimum arc length.

4 Press ENTER to continue with the command or ESC to end the command.

Toolbar: Draw 
Command entry: REVCLOUD

To edit the individual lengths of arcs or chords in a revision cloud

1 In your drawing, select the revision cloud you want to edit.

2 Move the pick points along the path of the revision cloud to change the arc lengths and chords.

Quick Reference

Commands

REVCLOUD

Creates a revision cloud using a polyline.

System Variables

DIMSCALE

Sets the overall scale factor applied to dimensioning variables that specify sizes, distances, or offsets.

DELOBJ

Controls whether geometry used to create 3D objects is retained or deleted.

Utilities

No entries

Command Modifiers

No entries

Create and Use Blocks (Symbols)

18

A block is one or more objects combined to create a single object. Blocks help you reuse objects in the same drawing or in other drawings.

Overview of Blocks

A block can be composed of objects drawn on several layers with various colors, linetypes, and lineweight properties. Although a block is always inserted on the current layer, the block reference preserves information about the original layer, color, and linetype properties of the objects that are contained in the block. You can control whether objects in a block retain their original properties or inherit their properties from the current layer, color, linetype, or lineweight settings.

Create Blocks

You can use several methods to create blocks:

- Combine objects to create a block definition in your current drawing.
- Use the Block Editor contextual tab (when the ribbon is active) or Block Editor (when the ribbon is not active) to add dynamic behavior to a block definition in your current drawing.
- Create a drawing file and later insert it as a block in other drawings.
- Create a drawing file with several related block definitions to serve as a block library.

You can use PURGE to remove unused block definitions from a drawing.

Dynamic Blocks

A block definition can contain elements that add dynamic behavior to a block, which adds flexibility and intelligence to the geometry. When you insert a block reference with dynamic behavior in a drawing, you can manipulate the geometry of the block reference through custom grips or custom properties, depending on how the block was defined. You can also constrain block geometry.

Annotative Blocks

You can also create blocks. For more information about creating and working with an annotative blocks, see Create Annotative Blocks and Attributes on page 1417.

See also:

- Scale Annotations on page 1393

Quick Reference

Commands

BLOCK

Creates a block definition from selected objects.

PURGE

Removes unused items, such as block definitions and layers, from the drawing.

WBLOCK

Writes objects or a block to a new drawing file.

System Variables

MAXSORT

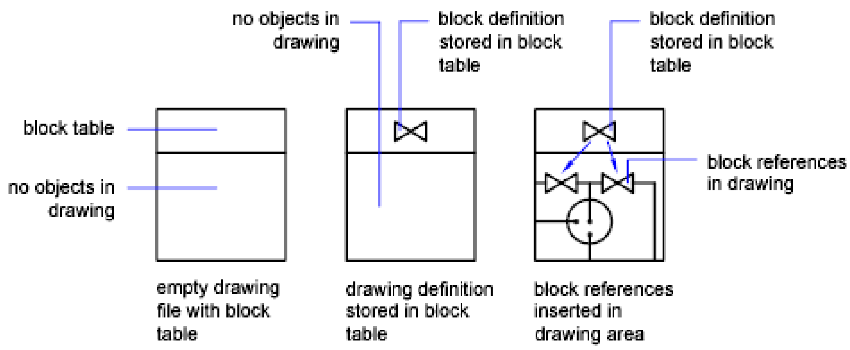
Sets the maximum number of symbol names or block names sorted by listing commands.

Create and Store Blocks

You create blocks by associating objects and giving them a name. You can also attach information (attributes) to a block.

How Blocks Are Stored and Referenced

The following illustrations are conceptual representations of three drawing files. Each rectangle represents a separate drawing file and is divided into two parts: the smaller part represents the block definition table, and the larger part represents the objects in a drawing.



When you insert a block, you are inserting a block reference. The information is not simply copied from the block definition to the drawing area. Instead, a link is established between the block reference and the block definition. Therefore, if the block definition is changed, all references are updated automatically.

To reduce the size of a drawing, you can purge unused block definitions.

Quick Reference

Commands

BLOCK

Creates a block definition from selected objects.

PURGE

Removes unused items, such as block definitions and layers, from the drawing.

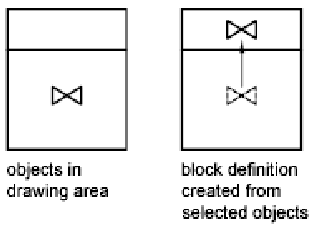
Create Blocks Within a Drawing

After you define a block in a drawing, you can insert a block reference in the drawing as many times as necessary. Use this method to create blocks quickly.

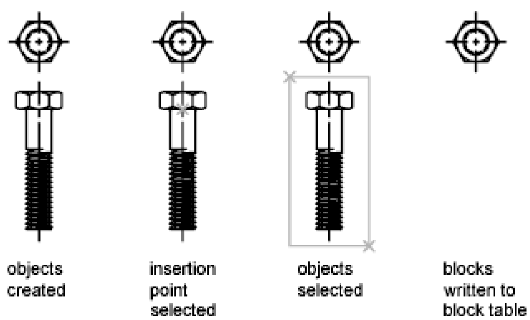
Each block definition includes a block name, one or more objects, the coordinate values of the base point to be used for inserting the block, and any associated attribute data.

The base point is used as a reference for positioning the block when you insert it. Suppose you specify that the base point is at the lower-left corner of an object in the block. Later, when you insert the block, you are prompted for an insertion point. The block base point is aligned at the insertion point you specified.

The block definition in the illustration comprises a name, PLUG_VALVE, four lines, and a base point at the intersection of the two diagonal lines. For an explanation of the schematic representation shown, see Overview of Blocks on page 853.



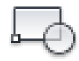
The illustration shows a typical sequence for creating a block definition within a drawing.



You can also use the Block Editor to create blocks that are saved within a drawing. For more information about using the Block Editor, see Use the Block Editor on page 868.

To define a block for the current drawing

1 Create the objects you want to use in the block definition.

2 Click Insert tab ➤ Block panel ➤ Create. 

3 In the Block Definition dialog box, enter a block name in the Name box.

4 Under Objects, select Convert to Block.

If you want the original objects used to create the block definition to remain in your drawing, make sure the Delete option is not selected. If this option is selected, the original objects are erased from the drawing. If necessary, you can use OOPS to restore them.

5 Click Select Objects.

6 Use your pointing device to select the objects to be included in the block definition. Press Enter to complete object selection.

7 In the Block Definition dialog box under Base Point, specify the block insertion point using one of these methods.

- Click Pick Point to specify a point using the pointing device.
- Enter the *X,Y,Z* coordinate values of the point.

8 In the Description box, enter a description for the block definition. This description is displayed in DesignCenter™ (ADCENTER).

9 Click OK.

The block is defined in the current drawing and can be inserted at any time.

Toolbar: Draw 
Command entry: BLOCK

Quick Reference

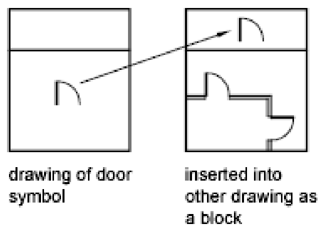
Commands

BLOCK

Creates a block definition from selected objects.

Create Drawing Files for Use as Blocks

You can create drawing files for the purpose of inserting them into other drawings as blocks. Individual drawing files are easy to create and manage as the source of block definitions. Collections of symbols can be stored as individual drawing files and grouped in folders.



Create a New Drawing File

You have two methods for creating drawing files:

- Create and save a complete drawing file using SAVE or SAVEAS.
- Create and save only selected objects from your current drawing to a new drawing using EXPORT or WBLOCK.

With either method, you create an ordinary drawing file that can be inserted as a block into any other drawing file. Using WBLOCK is recommended when you need to create several versions of a symbol as separate drawing files, or when you want to create a drawing file without leaving the current drawing.

Change the Base Point of Drawings to Be Used as Blocks

By default, the WCS (world coordinate system) origin (0,0,0) is used as the base point for drawing files inserted as blocks. You can change the base point by opening the original drawing and using BASE to specify a different base point for insertion. The next time you insert the block, the new base point is used.

Update Changes in the Original Drawing

If you change the original drawing after inserting it, the changes have no effect on the current drawing. If you expect the original drawing to change, and you want the changes to be reflected in the current drawing, you may want to attach it as an external reference instead of inserting it as a block. For more information about external references, see Reference Other Drawing Files.

Use Paper Space Objects in Blocks

Objects in paper space are not included when you insert a drawing as a block. To transfer paper space objects to another drawing, make the objects into a block or save them in a separate drawing file, and then insert the block or drawing file into the other drawing.

To create a new drawing file from selected objects

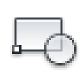
1. Open an existing drawing or create a new drawing.
2. At the Command prompt, enter **wblock**.
3. In the Write Block dialog box, select Objects.

   If you want the original objects used to create the new drawing to remain in your drawing, make sure the Delete From Drawing option is not selected. If this option is selected, the original objects are erased from the drawing. If necessary, you can use OOPS to restore them.
4. Click Select Objects.
5. Use your pointing device to select the objects to be included in the new drawing. Press Enter to complete object selection.
6. In the Write Block dialog box under Base Point, specify the point to be the origin point (0,0,0) for the new drawing using one of these methods:
   - Click Specify Point to specify a point using the pointing device.
   - Enter the *X,Y,Z* coordinate values of the point.
7. Under Destination, enter a file name and path for the new drawing, or click the [...] button to display a standard file selection dialog box.
8. Click OK.

   A new drawing is created with the selected objects.

Command entry: WBLOCK

To create a new drawing file from an existing block definition

1. Click Insert tab ➤ Block panel ➤ Create. 

2 In the Block Definition dialog box, in the Name box, select the block to modify.

3 In the Name box, enter a new name.

4 In the Description box, enter or modify the description for the new drawing file. Click OK.

Command entry: BLOCK

Quick Reference

Commands

ADCENTER

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

BASE

Sets the insertion base point for the current drawing.

BLOCK

Creates a block definition from selected objects.

EXPORT

Saves the objects in a drawing to a different file format.

INSERT

Inserts a block or drawing into the current drawing.

OOPS

Restores erased objects.

WBLOCK

Writes objects or a block to a new drawing file.

Control the Color and Linetype Properties in Blocks

Generally when you insert a block, the color, linetype, and lineweight of objects in the block retain their original settings regardless of the current settings in the drawing. However, you can create blocks with objects that inherit the current color, linetype, and lineweight settings. These objects have floating properties.

You have three choices for how the color, linetype, and lineweight properties of objects are treated when a block reference is inserted.

- Objects in the block do not inherit color, linetype, and lineweight properties from the current settings. The properties of objects in the block do not change regardless of the current settings.
  For this choice, it is recommended that you set the color, linetype, and lineweight properties individually for each object in the block definition: do not use BYBLOCK or BYLAYER color, linetype, and lineweight settings when creating these objects.

- Objects in the block inherit color, linetype, and lineweight properties from the color, linetype, and lineweight assigned to the current layer only.
  For this choice, before you create objects to be included in the block definition, set the current layer to 0, and set the current color, linetype, and lineweight to BYLAYER.

- Objects inherit color, linetype, and lineweight properties from the current color, linetype, and lineweight that you have set explicitly, that is, that you have set to override the color, linetype, or lineweight assigned to the current layer. If you have not explicitly set them, then these properties are inherited from the color, linetype, and lineweight assigned to the current layer.
  For this choice, before you create objects to be included in the block definition, set the current color or linetype to BYBLOCK.

If you want objects in a block to	Create objects on these layers	Create objects with these properties
Retain original properties	Any but 0 (zero)	Any but BYBLOCK or BYLAYER
Inherit properties from the current layer	0 (zero)	BYLAYER
Inherit individual properties first, then layer properties	Any	BYBLOCK

Floating properties also apply to nested blocks when the nested block references and the objects they contain use the settings required for floating properties.

To set the color for all new objects

1 Click Home tab ➤ Properties panel ➤ Object Color. 

2 Either click a color to draw all new objects in that color, or click Select Color to display the Select Color dialog box and do one of the following:
   - On the Index tab, click a color or enter the ACI color number (1-255) or name in the Color box, and then click OK.
   - On the True Color tab, select the HSL color model in the Color Model option and specify a color by entering a color value in the Color box or by specifying values in the Hue, Saturation, and Luminance boxes, and then click OK.
   - On the Color Books tab, select a color book from the Color Book box, select a color by navigating the color book (using the up and down arrows) and clicking on a color chip, and then click OK.
   - Click BYLAYER to draw new objects in the color assigned to the current layer.
   - Click BYBLOCK to draw new objects in the current color until they are grouped into a block. When the block is inserted into the drawing, the objects in the block acquire the current color setting.

3 Click OK.

   The Color control displays the current color.

Command entry: COLOR

To set the linetype for all new objects

1 Click Home tab ➤ Properties panel ➤ Linetypes. 

2 If you need to load additional linetypes, click Load, select one or more linetypes, and click OK.

   You can hold down Ctrl to select several linetypes or Shift to select a range of linetypes.

3 In the Linetype Manager, do one of the following:
   - Select a linetype and select Current to draw all new objects with that linetype.

- Select BYLAYER to draw new objects in the linetype assigned to the current layer.
- Select BYBLOCK to draw new objects in the current linetype until they are grouped into a block. When the block is inserted in a drawing, the objects in the block acquire the current linetype setting.

4 Click OK.

The Linetype control displays the current linetype. If the linetype you want to use is already loaded, you can click the Linetype control and click the linetype to make it current.

Command entry: LINETYPE
Menu: Click Format menu ➤ Linetype.

Quick Reference

Commands

COLOR

Sets the color for new objects.

LAYER

Manages layers and layer properties.

LINETYPE

Loads, sets, and modifies linetypes.

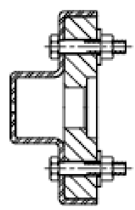
Nest Blocks

With nested blocks, you can build a single block out of several components. For example, you can insert as a block a drawing of a mechanical assembly that contains a housing, a bracket, and fasteners in which each fastener is a block composed of a bolt, washer, and nut. The only restriction on nested blocks is that you cannot insert blocks that reference themselves.

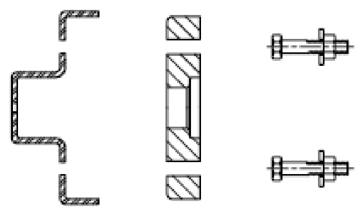
You can apply geometric constraints and constraint parameters to nested objects in blocks. AutoCAD detects the nested entity or valid constraint point for the nested entity regardless of the nesting level of the object.

NOTE Constraints can only be applied between nested objects in the block and objects in the drawing file, not between pairs of nested objects in the block reference.

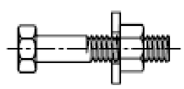
When a block definition is redefined, AutoCAD will re-evaluate the constraints between geometry in the drawing and the nested geometry in the block references. The drawing will then be updated appropriately. If a constraint cannot be resolved as a result of the change to the block definition, then the constraint is removed and an unresolved constraints message is displayed at the command line.



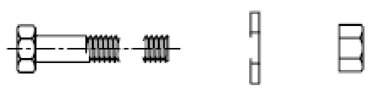
assembly block



blocks that are components of the assembly block



fastener block



blocks that are components of the fastener block

Quick Reference

Commands

BLOCK

Creates a block definition from selected objects.

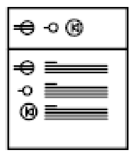
Create Block Libraries

A block library is a collection of block definitions stored in a single drawing file. You can use block libraries supplied by Autodesk or other vendors or create your own.

You can organize a set of related block definitions by creating the blocks in the same drawing file. Drawing files used this way are called block, or symbol, libraries. These block definitions can be inserted individually into any drawing that you are working on. Block library drawings are not different from other drawing files except in how they are used.

When you use BLOCK to define each block definition in the block library drawing, you can include a short description of the block that can be viewed in DesignCenter.

Optionally, you can also document each block definition by inserting it in the drawing area of the library drawing. In addition to the block geometry, you can include text that provides the block name, the date of creation, the date of the last modification, and any special instructions or conventions. This creates a visual index of the blocks in the block library drawing.



sample block
library drawing

Use DesignCenter to view and copy block definitions individually from block library drawings (or from any existing drawing) to your current drawing. DesignCenter does not overwrite an existing block definition in a drawing with one that comes from another drawing.

To create a block library drawing

1. Begin a new drawing.
2. Define a block.
3. Repeat step 2 for as many related block definitions as you want to make.
4. Save the drawing using a name appropriate for a library drawing.

   These blocks can be inserted into any drawing using DesignCenter (ADCENTER).

Toolbar: Draw 
Command entry: BLOCK

Quick Reference

Commands

BLOCK

Creates a block definition from selected objects.

Use Tool Palettes to Organize Blocks

You can use tool palettes to organize blocks that are stored in one drawing file or separate drawing files.

Once you've added a block tool to a tool palette, you can easily insert the block reference in your drawing by dragging it from the tool palette to the drawing or by clicking and placing it in the drawing. For information about using tool palettes to organize and insert blocks, see Create and Use Tools from Objects and Images on page 56.

Quick Reference

Commands

TOOLPALETTES

Opens the Tool Palettes window.

Remove Block Definitions

To reduce the size of a drawing, you can remove unused block definitions. You can remove a block reference from your drawing by erasing it; however, the block definition remains in the drawing's block definition table.

To remove unused block definitions and decrease the drawing size, use PURGE at any time in your drawing session.

All references to a block must be erased before you can purge the block definition.

See also:

- Overview of Blocks on page 853

To remove a block definition

1 Click Tools tab ➤ Drawing Utilities panel ➤ Purge. 

The Purge dialog box displays a tree view of named objects that can be purged.

2 To purge blocks, use one of the following methods:

- To purge all unreferenced blocks, select Blocks. To include nested blocks, select Purge Nested Items.
- To purge specific blocks, double-click Blocks to expand the Block tree view. Select the blocks to be purged.

If the item you want to purge is not listed, select View Items You Cannot Purge.

3 You are prompted to confirm each item in the list. If you do not want to confirm each purge, clear the Confirm Each Item to Be Purged option.

4 Click Purge.

To confirm the purging of each item, respond to the prompt by choosing Yes or No, or Yes to All if more than one item is selected.

5 Select more items to purge, or click Close.

Command entry: PURGE

Quick Reference

Commands

PURGE

Removes unused items, such as block definitions and layers, from the drawing.

Add Dynamic Behavior to Blocks

Instead of being a fixed part of a drawing, a dynamic block reference can be changed or manipulated as you work in a drawing.

Use the Block Editor

You use the Block Editor to add dynamic behavior to blocks.

Overview of the Block Editor

The Block Editor is a special authoring area for creating block definitions and adding dynamic behavior.

The Block Editor provides quick access to block authoring tools.

The Block Editor provides a drawing area in which you can draw and edit geometry as you would in the program's main drawing area. You can specify the background color for the Block Editor drawing area in the Display tab of the Options dialog box.

NOTE You can use most commands in either the Block Editor contextual tab or the Block Editor. When you enter a command that is not allowed in the Block Editor, a message is displayed at the Command prompt.

You can use either the Block Editor contextual tab or the Block Editor to edit or add dynamic behavior to block definitions that exist in the current drawing. You can also use it to create new block definitions.

When the ribbon is active, a special ribbon contextual tab is displayed above the drawing area. When the ribbon is not active, a special toolbar is displayed. Both the ribbon contextual tab and toolbar provide tools to do the following:

- Add a constraint
- Add a parameter
- Add an action
- Define attributes
- Close the Block Editor
- Manage visibility states
- Save the block definition

You can select any parameter, grip, action, or geometric object in either the Block Editor contextual tab or the Block Editor to view its properties in the Properties palette. When you select an object in the Block Editor, the coordinate values shown in the Properties palette reflect the block definition space.

When you work in the Block Editor contextual tab or the Block Editor, the command line should be displayed. The command line displays prompts for nearly all aspects of creating dynamic blocks.

UCS in the Block Editor

A UCS icon is displayed in the drawing area of the Block Editor. The origin of the UCS icon defines the base point for the block. You can change the base point for the block by moving the geometry relative to the origin of the UCS icon, or by adding a base point parameter.

The UCS command is disabled in the Block Editor. You can open an existing 3D block definition in the Block Editor and assign parameters to the block. However, the parameters will ignore any *Z* coordinate values in the block space. Consequently, the block reference cannot be edited along the *Z* axis. Furthermore, while you can create a dynamic block that contains solid objects and add actions to it, such as move, rotate, and scale, you can't perform solid editing features within a dynamic block reference (for example, stretch a solid, move a hole within a solid, and so on).

See also:

- Define Block Attributes on page 1029

To open an existing block definition in the Block Editor

1 Click Insert tab ➤ Block panel ➤ Edit. 

2 In the Edit Block Definition dialog box, do one of the following:
   - Select a block definition from the list.
   - Select <Current Drawing> if the drawing is the block definition you want to open.

3 Click OK.

Toolbar: Standard 
Command entry: BEDIT
Shortcut menu: Right-click a selected block. Click Block Editor.

To create a new block definition in the Block Editor

1 Click Insert tab ➤ Block panel ➤ Edit.

2 In the Edit Block Definition dialog box, enter a name for the new block definition. Click OK.

3 Click Block Editor tab ➤ Open/Save panel ➤ Save Block.

   NOTE This saves the block definition even if you have not added any objects in the Block Editor drawing area.

4 Click Close Block Editor.

Toolbar: Standard
Command entry: BEDIT

To open a block on a tool palette in the Block Editor

1 If the Tool Palettes window is not already open, click View tab ➤ Palettes panel ➤ Tool Palettes.

2 Right-click a block icon.

3 Click Block Editor.

NOTE A block on a tool palette may reside in another drawing. The drawing that contains the block definition is opened in the Block Editor.

Command entry: TOOLPALETTES

Toolbar: Standard

To open a block from the DesignCenter window in the Block Editor

1 Click View tab ➤ Palettes panel ➤ DesignCenter. 
2 Right-click a block icon.
3 Click Block Editor.

Command entry: ADCENTER

Toolbar: Standard 

To open a drawing file saved as a block (not dynamic) in the Block Editor

1 Click the Application button, and click Open ➤ Drawing.
2 Open the drawing file that is saved as a block.
3 Click Insert tab ➤ Block panel ➤ Edit. 
4 In the Edit Block Definition dialog box, select <Current Drawing>. Click OK.

Toolbar: Standard 
Command entry: OPEN

To open a drawing file saved as a dynamic block in the Block Editor

1 Click the Application button, and click Open ➤ Drawing.
2 Open the drawing file that is saved as a block.
   An alert is displayed that states that the drawing file contains authoring elements.
3 In the alert dialog box, click Yes to open the drawing in the Block Editor.

Command entry: OPEN

Toolbar: Standard 

To view properties of a block definition in the Block Editor

1 Click Insert tab ➤ Block panel ➤ Edit.

2 In the Edit Block Definition dialog box, do one of the following:
   - Select a block definition from the list.
   - Select <Current Drawing> if the drawing is the block definition you want to open.

3 Click OK.

4 Click View tab ➤ Palettes panel ➤ Properties.

5 In the Properties Palette window, under Block, view the properties of the block definition.

Toolbar: Standard
Command entry: BEDIT

To view properties of objects in the Block Editor

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 In the Block Editor, select an object.

3 Click View tab ➤ Palettes panel ➤ Properties.

4 In the Properties Palette window, view the properties of the selected object.

Command entry: BEDIT

Toolbar: Standard
Shortcut menu: Right-click the selected object. Click Properties.

To close the Block Editor

- Click Block Editor tab ➤ Close panel ➤ Close Block Editor.

Command entry: BCLOSE

To prevent access to the Block Editor

1 At the Command prompt, enter **blockeditlock**.

2 Enter **1**, and then press Enter.

Quick Reference

Commands

BCLOSE

Closes the Block Editor.

BEDIT

Opens the block definition in the Block Editor.

OPTIONS

Customizes the program settings.

PROPERTIES

Controls properties of existing objects.

System Variables

BLOCKEDITLOCK

Disallows opening of the Block Editor and editing of dynamic block definitions.

Use the Parameters Manager

The Parameters Manager allows you to display and edit constraints, user parameters, user variables, action parameters, and block attributes from within the Block Editor.

Within the Block Editor, the Parameters Manager displays and controls the following categories:

- Constraint Parameters
- User Parameters
- Action Parameters
- Dimensional and Reference Constraints
- User Variables
- Block Attributes

For each of the above categories, you can display and control the following:

- Expression
- Value
- Type
- Display Order
- Show or Hide Information
- Description

For more information on the Parameters Manager in the Block Editor, see Parameters Manager - Dynamic Blocks.

To display the Parameters Manager in the Block Editor

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. Click Block Editor tab ➤ Manage panel ➤ Parameters Manager.

Command entry: PARAMETERS

To display or hide additional columns in the Parameters Manager

- In the Parameters Manager, right-click a column header and select or de-select a column name.

Quick Reference

Commands

PARAMETERS

Controls the associative parameters used in the drawing.

PARAMETERSCLOSE

Closes the Parameters Manager palette.

System Variables

PARAMETERSSTATUS

Indicates whether the Parameters Manager palette is displayed or hidden.

Define User Parameters in Dynamic Blocks

You can control the geometry of constraint parameters with mathematical expressions.

While you can insert both user variables and *user parameters* in the Block Editor, only user parameters are displayed as editable custom properties for a block reference.

There are several types of user parameters. For more information, see Parameters Manager - Dynamic Blocks.

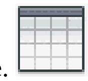
NOTE User parameters can be exposed as custom properties of the block reference. If it does not define an equation, then the property can be edited. If it is an equation, then the property is read-only.

See also:

- Parameters Manager - Dynamic Blocks
- Constrain a Design with Formulas and Equations on page 758
- Specify Custom Properties for Dynamic Blocks on page 1004

To create a new user parameter

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 Click Block Editor tab ➤ Dimensional panel ➤ Block Table. 

3 In the Block Properties Table dialog box, click New Properties.

4 In the New Parameter dialog box, specify a name and a value. Click OK.

Command entry: BTABLE

To create user parameters using the Parameters Manager

➤

To change the type of user parameter

➤ Use parameters manager

Quick Reference

Commands

BCPARAMETER

Applies constraint parameters to selected objects, or converts dimensional constraints to parameter constraints.

BTABLE

Displays a dialog box to define variations of a block.

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects.

System Variables

BCONSTATUSMODE

Turns the constraint display status on and off and controls the shading of objects based on their constraint level.

Use a Block Properties Table

You can define and control values for parameters and properties within a block definition using a Block Properties table.

The Block Properties Table dialog box consists of a grid with parameters defining column headings and rows defining the different property set values. When selecting a block reference, it can be set to the values defined by one of the rows in the block properties table.

A table can include any of the following parameters and properties:

- Action parameters
- User parameters
- Constraint parameters
- Attributes

NOTE Only one table is allowed in the block definition.

You can restrict the property set for block references to only values defined by one of the rows of the table. You can also change individual properties of the reference so that the property set no longer matches one of the defined rows.

To add parameter properties to a block table

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. In the Block Editor, double-click a block properties table in the block definition.
3. In the Block Properties Table dialog box, click Add Properties.
4. In the Add Parameter Properties dialog box, under the Parameter Properties list, select the parameter properties you want to add to the block table. Hold Ctrl to select more than one property.
5. Click OK.

Command entry: BTABLE

To check for errors in a block properties table

1. Click Block Editor tab ➤ Dimensional panel ➤ Block Table.
2. In the Block Properties Table dialog box, click Audit.

Command entry: BTABLE

To open an existing block properties table

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 In the Block Editor, double-click a block properties table.

Command entry: BTABLE

Quick Reference

Commands

BCPARAMETER

Applies constraint parameters to selected objects, or converts dimensional constraints to parameter constraints.

BTABLE

Displays a dialog box to define variations of a block.

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects.

System Variables

BCONSTATUSMODE

Turns the constraint display status on and off and controls the shading of objects based on their constraint level.

Use Block Authoring Palettes

The Block Editor has a block authoring palette with four tabs: Parameters, Actions, Parameter Sets, and Constraints.

The Block Authoring Palettes window is displayed only in the Block Editor. Use these palettes to add parameters and actions to your dynamic block definition.

Create Custom Block Authoring Tools

You can create custom block authoring tools. In order to preserve the default tools on the block authoring palettes, you should create a new palette for custom block authoring tools. You can then copy a parameter set tool from one of the existing palettes and paste the copy onto the new palette. In the Tool Properties dialog box, you can then change the properties of the new tool, including the following:

- Tool description
- Parameter type
- Associated action(s)
- Key point on the parameter to which the action is tied (if applicable)
- Tool palette image

You cannot drag parameters and actions from the Block Editor onto any tool palette.

See also:

- Add Action Parameters to Dynamic Blocks on page 936
- Add Actions to Dynamic Blocks on page 960
- Use Parameter Sets on page 984
- Control Tool Properties on page 70
- Organize Tool Palettes on page 80

To show or hide the Block Authoring palettes in the Block Editor

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. In the Edit Block Definition dialog box, Under Block To Create or Edit, select a name from the list and then, click OK.
3. Click Block Editor tab ➤ Manage panel ➤ Authoring Palettes. 

Toolbar: Standard

Command entry: BAUTHORPALETTE, BAUTHORPALETTECLOSE

To make a copy of a parameter set

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 In the Edit Block Definition dialog box, Under Block To Create or Edit, select a name from the list and then, click OK.

3 Click Block Editor tab ➤ Manage panel ➤ Authoring Palettes.

4 In the Block Authoring Palettes window, Parameters Sets tab, right-click a parameter set. Click Copy.

5 Right-click anywhere on the palette to which you want to add the parameter set (except on a parameter set). Click Paste.

To add an action to a parameter set

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 In the Edit Block Definition dialog box, Under Block To Create or Edit, select a name from the list and then, click OK.

3 Click Block Editor tab ➤ Manage panel ➤ Authoring Palettes.

4 In the Block Authoring Palettes window, Parameters Sets tab, right-click a parameter set. Click Properties.

5 In the Tool Properties dialog box, under Parameter, click Actions, and then click the [...] button.

6 In the Add Actions dialog box, under Action Object to Add, select an action from the list.

7 Click Add.

8 (Optional) Repeat steps 3 and 4 to add additional actions.

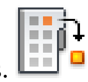
9 Click OK.

10 In the Tool Properties dialog box, click OK.

To delete an action from a parameter or parameter set

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 In the Edit Block Definition dialog box, Under Block To Create or Edit, select a name from the list and then, click OK.

3 Click Block Editor tab ➤ Manage panel ➤ Authoring Palettes. 

4 In the Block Authoring Palettes window, right-click a parameter or parameter set. Click Properties.

5 In the Tool Properties dialog box, under Parameter, click Actions, and then click the [...] button.

6 In the Add Actions dialog box, under Action Object List, select an action from the list.

7 Click Delete.

8 (Optional) Repeat steps 3 and 4 to delete additional actions.

9 Click OK.

10 In the Tool Properties dialog box, click OK.

Command entry: BEDIT ➤ BAUTHORPALETTE

Quick Reference

Commands

BAUTHORPALETTE

Opens the Block Authoring Palettes window in the Block Editor.

BAUTHORPALETTECLOSE

Closes the Block Authoring Palettes window in the Block Editor.

BEDIT

Opens the block definition in the Block Editor.

TOOLPALETTES

Opens the Tool Palettes window.

System Variables

BLOCKEDITOR

Indicates whether or not the Block Editor is open.

How Objects Are Displayed in the Block Editor

Parameters, actions, and their relationships (dependencies) are displayed in different ways in the Block Editor. You can specify settings for some of these elements.

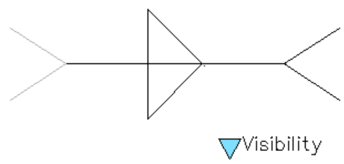
Parameters

In the Block Editor, most parameters look like dimensions. If you create a value set (a range or list of values) for a parameter, tick marks are shown at the locations of those values.

You can specify the following settings for parameters in the Block Editor:

- Parameter color
- Parameter text and arrow size
- Parameter font
- Grip color
- Display of value set markers (tick marks) for parameters

When you use a visibility parameter in your dynamic block definition, you can specify which geometric objects are invisible for a given visibility state. You can specify whether or not geometry that is made invisible for visibility states displays in the Block Editor. In the following example, a visibility state is displayed in the Block Editor. The geometry that displays in a dimmed state is invisible for that visibility state.



Actions

An action displays its name and icon (a lightning bolt) in the Block Editor. You can specify the text size and color for actions in the Block Editor.

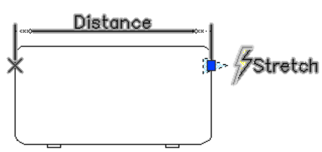
Grips

You can specify grip size and color for display in the Block Editor. This setting does not affect the size and color of the grips in a block reference in a drawing.

Dependencies

When you select a parameter, grip, or action in the Block Editor, its associated objects, or dependencies, are highlighted. This is called *dependency highlighting*. You can turn dependency highlighting on or off.

The following example shows how dependency highlighting creates a halo effect for the associated parameter (labeled Distance) and action (labeled Stretch) when you select the custom grip in the Block Editor.



The following table details what is dependency highlighted when you select an element in the Block Editor.

Selected object in the Block Editor	Objects that are dependency highlighted
Parameter	Associated grips and actions
Grip	Associated parameter and actions
Action	Associated parameters, grips, and the selection set (geometry)

To specify the text size for parameters and actions in the Block Editor

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 Click Block Editor tab ➤ Manage panel ➤ Block Editor Settings.

3 In the Block Editor Settings dialog box, under Parameter and Grip Size, specify the parameter size.

4 Enter an integer from 1 to 255 (pixels). Click OK.

Command entry: BESETTINGS

To specify the font for parameters in the Block Editor

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 Click Block Editor tab ➤ Manage panel ➤ Block Editor Settings.

3 In the Block Editor Settings dialog box, under Parameter Font, select the font name and font style. Click OK.

Command entry: BESETTINGS

To specify the text color for actions in the Block Editor

1 At the Command prompt, enter **bactioncolor**.

2 Enter one of the following values:

- **BYLAYER**
- **BYBLOCK**
- An integer from 1 to 255.
- A true color specified by three integers each ranging from 1 to 255 in the following format: **RGB:000,000,000**.

3 Press Enter.

To specify the display size of grips in the Block Editor

1. Click Insert tab ➤ Block panel ➤ Block Editor.
2. Click Block Editor tab ➤ Manage panel ➤ Dialog box launcher.
3. In the Block Editor Settings dialog box, under Parameter and Grip Size, specify the grip size.
4. Enter an integer from 1 to 255 (pixels). Click OK.

Command entry: BESETTINGS

To specify the display color of grips in the Block Editor

1. Click Insert tab ➤ Block panel ➤ Block Editor.
2. Click Block Editor tab ➤ Manage panel ➤ Dialog box launcher.
3. In the Block Editor Settings dialog box, under Authoring Objects, select the grip color. Click OK.

Command entry: BESETTINGS

To specify the parameter text alignment in the Block Editor

1. Click Insert tab ➤ Block panel ➤ Block Editor.
2. Click Block Editor tab ➤ Manage panel ➤ Dialog box launcher.
3. In the Block Editor Settings dialog box, under Authoring Objects, specify the parameter text alignment. Click OK.

Command entry: BESETTINGS

To update the text and grip display size at their specified values in the Block Editor

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 In the Block Editor, on the Block Editor toolbar, click Update Parameter and Action Text Size.

NOTE When you zoom in and out in the Block Editor, text and grip size will change relative to the zoom factor. However, you can update the Block Editor to display the text and grip size at their specified values.

Command entry: REGEN

To specify whether or not objects that are invisible for a visibility state can be seen in the Block Editor

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 Click Block Editor tab ➤ Visibility panel ➤ Visibility Mode.

Command entry: BVMODE

To turn dependency highlighting on or off in the Block Editor

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 Click Block Editor tab ➤ Manage panel ➤ Dialog box launcher.

3 Select Highlight Dependent Objects during Selection.

4 Click OK.

Command entry: BESETTINGS

To control whether value set markers (tick marks) are displayed

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 Click Block Editor tab ➤ Manage panel ➤ Dialog box launcher. 

3 Select Display Tickmarks for Parameters with Value Sets. Click OK.

Command entry: BESETTINGS

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BESETTINGS

Displays the Block Editor Settings dialog box.

REGEN

Regenerates the entire drawing from the current viewport.

System Variables

BACTIONCOLOR

Sets the text color of actions in the Block Editor.

BDEPENDENCYHIGHLIGHT

Controls whether or not dependent objects are dependency highlighted when a parameter, action, or grip is selected in the Block Editor.

BGRIPOBJCOLOR

Sets the color of grips in the Block Editor.

BGRIPOBJSIZE

Sets the display size of custom grips in the Block Editor relative to the screen display.

BPARAMETERCOLOR

Sets the color of parameters in the Block Editor.

BPARAMETERFONT

Sets the font used for parameters and actions in the Block Editor.

BPARAMETERSIZE

Sets the size of parameter text and features in the Block Editor relative to the screen display.

BPTEXTHORIZONTAL

Forces the text displayed for action parametes and constraint parameters in the Block Editor to be horizontal.

BTMARKDISPLAY

Controls whether or not value set markers are displayed for dynamic block references.

BVMODE

Controls how objects that are made invisible for the current visibility state are displayed in the Block Editor.

Modify Dynamic Block Definitions

After you've defined a dynamic block, you can modify it in the Block Editor. You can delete, add, and modify the following elements in the Block Editor:

- Constraints (see Add Constraints to Dynamic Blocks on page 898)
- Parameters (see Add Action Parameters to Dynamic Blocks on page 936)
- Actions (see Add Actions to Dynamic Blocks on page 960)
- Geometry
- Visibility states (see Create Visibility States)
- Lookup tables (see Use Lookup Tables to Assign Data to Dynamic Blocks on page 993)
- Grips (see Specify Grips for Dynamic Blocks on page 987)
- Properties (see Overview of Specifying Custom Properties for Dynamic Blocks on page 1005)

You can also use the BACTIONSET command to change an action's selection set. If you delete a parameter associated with an action and need to reassign that action to another parameter, use the BASSOCIATE command.

After you modify a dynamic block definition in the Block Editor, you must save your changes (see Save Blocks on page 896).

WARNING If you redefine a dynamic block in AutoCAD 2005 or earlier, the block will lose its dynamic behavior.

To open an existing dynamic block definition in the Block Editor

1. Open the drawing file that contains the dynamic block definition, or open the drawing file that has been saved as a block.
2. Follow the steps in To open an existing dynamic block definition in the Block Editor on page 889.

Toolbar: Standard 
Command entry: OPEN
Shortcut menu: Right-click a selected block. Click Block Editor.

To delete an element from a dynamic block definition

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. In the Block Editor, select the element you want to delete from the dynamic block definition.
3. Press Delete.

To modify an action's selection set in a dynamic block definition

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. In the Block Editor, select an action.
3. At the Command prompt, enter **bactionset**.
4. Follow the Command prompts.

5 Click Block Editor tab ➤ Open/Save panel ➤ Save Block.

6 (Optional) If you are finished using the Block Editor, click Close Block Editor.

To redefine a stretch or polar stretch action's stretch frame set in a dynamic block definition

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 In the Block Editor, double-click a stretch or polar stretch action.

3 Follow the Command prompts to define a new stretch frame and add objects to or remove objects from the action's selection set.

4 Click Block Editor tab ➤ Open/Save panel ➤ Save Block.

5 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Command entry: BACTIONSET

Quick Reference

Commands

BACTION

Adds an action to a dynamic block definition.

BACTIONTOOL

Adds an action to a dynamic block definition.

BACTIONSET

Specifies the selection set of objects associated with an action in a dynamic block definition.

BASSOCIATE

Associates an action with a parameter in a dynamic block definition.

BEDIT

Opens the block definition in the Block Editor.

BGRIPSET

Creates, deletes, or resets grips associated with a parameter.

BPARAMETER

Adds a parameter with grips to a dynamic block definition.

BLOOKUPTABLE

Displays or creates a lookup table for a dynamic block definition.

BVSTATE

Creates, sets, or deletes a visibility state in a dynamic block.

Correct Errors in Dynamic Block Definitions

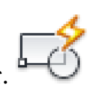
A yellow alert icon is displayed in the Block Editor if a dynamic block definition contains errors or is incomplete. You need to correct the errors (or complete the block) so the block reference will function properly in a drawing.

The yellow alert icon indicates that the block definition is not defined correctly or is incomplete. For example, a parameter that is not associated with an action will display an alert icon. An action that is not associated with a parameter or a selection set, will also display an alert icon.

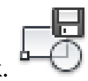
You can correct these errors by double-clicking the yellow alert icon and following the Command prompts.

After you create a dynamic block definition in the Block Editor, you should save it and then test the functionality of the block reference in a drawing. If the block reference isn't functioning the way you intended, open the definition in the Block Editor and double-check the types of parameters and actions you used as well as their dependencies and properties.

To correct an error indicated by a yellow alert icon in a dynamic block definition

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. In the Block Editor, double-click a yellow alert icon.

3 Follow the Command prompts to correct the errors in the dynamic block definition.

4 Click Block Editor tab ➤ Open/Save panel ➤ Save Block. 

5 (Optional) If you are finished using the Block Editor, click Close Block Editor.

To view dependencies within a dynamic block definition

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 In the Block Editor, select a parameter or an action.

If you selected a parameter, its associated action is highlighted.

If you selected an action, its associated parameter and selection set of geometry is highlighted.

3 When you're finished viewing the dependencies, press ESC.

4 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Quick Reference

Commands

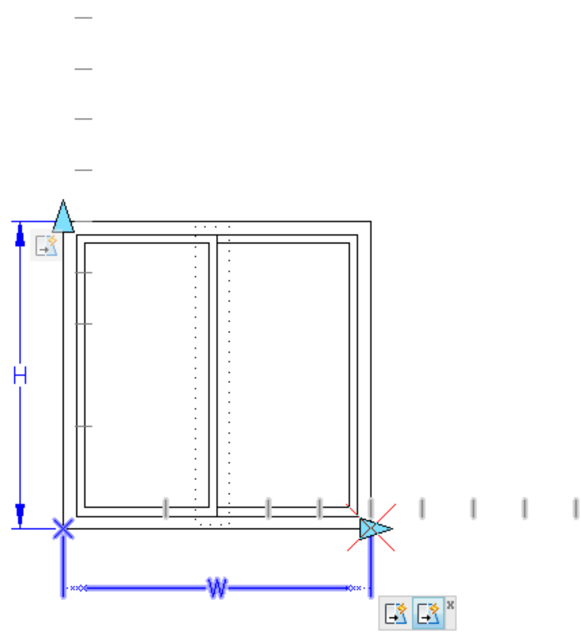
BEDIT

Opens the block definition in the Block Editor.

View Actions Related to a Parameter

Action objects are grouped into Action bars based on their associated parameters.

Action bars provide a quick visual representation of all actions associated with a parameter. Each action is displayed as an icon on the action bar.



The action bar is located relative to the parameter object by default but can be moved to any location.

Hover the cursor over an action icon to display

- The associated parameter
- The associated selection set
- Additional graphics that belong to the action

NOTE In order to use the BASSOCIATE command, you must first set BACTIONBARMODE to 0.

To display action bars for all parameters

- Click Insert tab ➤ Block panel ➤ Block Editor. 
- Click Block Editor tab ➤ Action Parameters panel ➤ Show All Actions. 

Command entry: BACTIONBAR

To hide action bars for all parameters

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 Click Block Editor tab ➤ Action Parameters panel ➤ Hide All Actions. 

Command entry: BACTIONBAR

To turn on or off the Action bar mode

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 Click Block Editor tab ➤ Manage panel ➤ Dialog box launcher. 

3 In the Block Editor Settings dialog box, select Display Action Bars. Click OK.

Command entry: BACTIONBARMODE

Quick Reference

Commands

BACTIONBAR

Displays or hides action bars for a selection set of parameter objects.

System Variables

BACTIONBARMODE

Indicates whether the action bars or the legacy action objects are displayed in the Block Editor.

Test Blocks Within the Block Editor

A test block window is displayed to allow easy testing of the block definition while creating dynamic blocks.

The block definition does not need to be saved to test the edits made in the Block Editor. The test block window reflects the current block definition in the Block Editor. You can test the block without exiting the Block Editor.

The test block window automatically closes when you exit the Block Editor, open a different block from the Block Editor, or save the current block definition to a different name.

Only one test block window can be open at a time. If a test block window is open and you enter BTESTBLOCK, the window closes and a new test block window is opened.

Most AutoCAD commands are unchanged in the test block window except the following commands:

- BEDIT - Disabled in test block window
- SAVE, SAVEAS, and QSAVE - Do not display a default file name in the Save dialog box. If you save from the test block window, the contextual tab is removed and a new drawing is created. The test block window is closed.
- CLOSE and QUIT- Do not prompt to save when closing the test block window.

To test a dynamic block

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. Click Block Editor tab ➤ Open/Save panel ➤ Test Block. 

Command entry: BTESTBLOCK

To close the test block window

- In the test block window, click Close Test Block Window. 
  Closing the test block window returns you to the Block Editor.

Quick Reference

Commands

BTESTBLOCK

Displays a window within the Block Editor to test a dynamic block.

System Variables

BLOCKTESTWINDOW

Indicates whether or not a test block window is current.

Save Blocks

When you are finished adding elements to your dynamic block definition in the Block Editor, save the block definition.

In the Block Editor, you can save your block definition by clicking the Save Block button on the Block Editor contextual tab, or by entering **bsave** at the Command prompt. You should then save your drawing to make sure that the block definition is saved in the drawing.

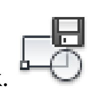
When you save a block definition in the Block Editor, the current values of the geometry and parameters in the block are set as the default values for the block reference. When you create a dynamic block that uses visibility states, the default visibility state for the block reference is the visibility state at the top of the list in the Manage Visibility States dialog box.

Once you've saved the block definition, you can close the Block Editor and try your block in a drawing.

NOTE If you click File menu ➤ Save while you are in the Block Editor, you will save the drawing but not the block definition. You must specifically save the block definition while you are in the Block Editor.

To save a block definition in the Block Editor

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 Click Block Editor tab ➤ Open/Save panel ➤ Save Block. 

3 To save the block definition in the drawing, click File menu ➤ Save.

4 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Command entry: BSAVE

To save a copy of the current block definition in the Block Editor under a new name

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 Click Block Editor tab ➤ Open/Save panel ➤ Save Block As. 

3 In the Save Block As dialog box, enter a name for the new block definition.

4 Click OK.

5 To save the block definition in the drawing, click File menu ➤ Save.

6 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Command entry: BSAVEAS

To save the current block definition as a new drawing file

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 Click Block Editor tab ➤ Open/Save panel ➤ Save Block As. 

3 In the Save Block As dialog box, enter a name for the new block definition.

4 Select the Save Block Definition to Drawing File check box.

5 Click OK.

6 In the Browse For Drawing File dialog box, click Save.

Command entry: BSAVEAS

Quick Reference

Commands

BCLOSE

Closes the Block Editor.

BEDIT

Opens the block definition in the Block Editor.

BSAVE

Saves the current block definition.

BSAVEAS

Saves a copy of the current block definition under a new name.

Add Constraints to Dynamic Blocks

You can add geometric constraints and constraint parameters to a dynamic block in the Block Editor.

Add Geometric Constraints to Dynamic Blocks

A geometric constraint defines a relationship between two objects, or between an object and a coordinate system.

Overview of Geometric Constraints in Dynamic Blocks

With geometric constraints, you can

- Maintain parallel, perpendicular, tangent, or coincidence points between two objects
- Force a line or pair of points to remain vertical or horizontal
- Fix a point on an object to the WCS

See also:

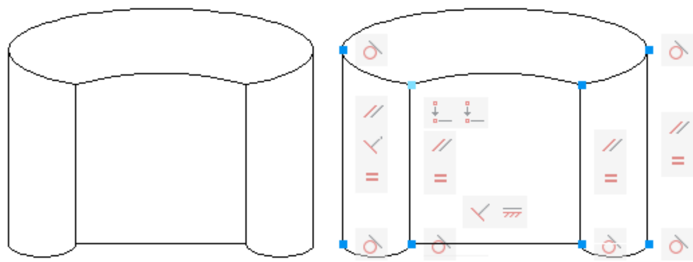
- Constrain Objects Geometrically on page 723

Automatically Constrain a Block Definition

You can automatically apply geometric constraints to the objects in the block definition.

With the AUTOCONSTRAIN command, constraints are applied to objects that already fit the criteria for geometric constraints as they are currently drawn in the block definition.

For example, in the illustration below, an armchair is depicted on the left. The AUTOCONSTRAIN command was applied to the armchair.



Because of the geometry of the objects, the following constraints were automatically applied to the armchair on the right:

- Coincident constraint on page 900
- Parallel constraint on page 905
- Perpendicular constraint on page 907
- Horizontal constraint on page 908
- Tangent constraint on page 911
- Equal constraint on page 915

See also:

- To apply multiple geometric constraints to an object on page 732

Quick Reference

Commands

AUTOCONSTRAIN

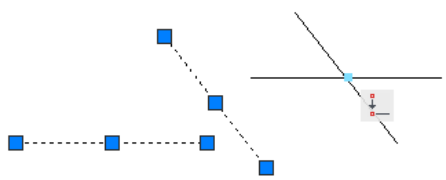
Applies geometric constraints to a selection set of objects based on orientation of the objects relative to one another.

Add a Coincident Constraint

A coincident constraint forces two points, or a point and line, to coincide.

Apply a Coincident Constraint

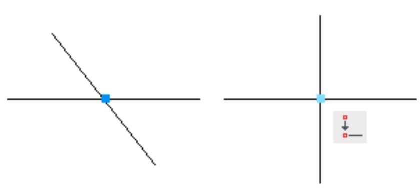
In the example below, a coincident constraint has been applied to the midpoints of the two lines on the left. The result is the block definition on the right, where the midpoints coincide with each other.



Manipulate a Coincident Constraint

When you manipulate a block with a coincident constraint applied, the specified points will always coincide with each other.

In the example below, the constrained block on the left has been manipulated to produce the block on the right. The two lines can move and change shape, but will always be constrained to coincide at their midpoints.



See also:

- Apply or Remove Geometric Constraints on page 725
- To apply a coincident constraint on page 728

Quick Reference

Commands

GEOMCONSTRAINT

Applies or persists geometric relationships between objects or points on objects.

Add a Colinear Constraint

A colinear constraint forces two lines to follow the same infinite line.

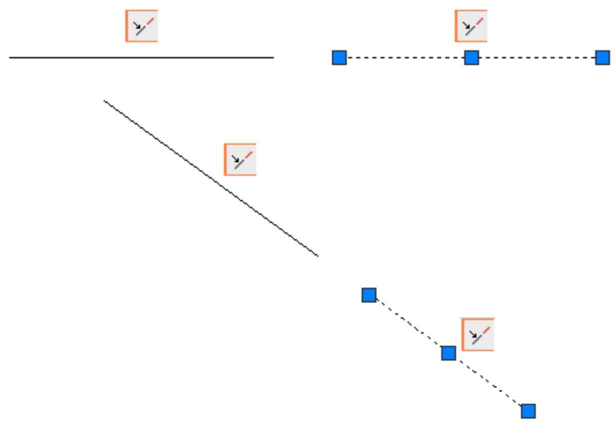
Apply a Colinear Constraint

In the example below, a colinear constraint has been applied to the two lines on the left. The result is the block definition on the right, where the two lines are aligned.

Manipulate a Colinear Constraint

When you manipulate a block with a colinear constraint applied, the specified lines will always be aligned with each other.

In the example below, the constrained lines on the top have been rotated to produce the block on the bottom. The two lines can move, change angle, or change shape, but will always be constrained to align.



See also:

- Apply or Remove Geometric Constraints on page 725
- To apply a collinear constraint on page 728

Quick Reference

Commands

GEOMCONSTRAINT

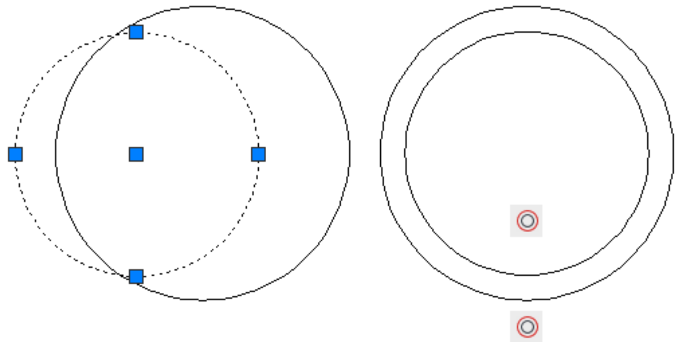
Applies or persists geometric relationships between objects or points on objects.

Add a Concentric Constraint

A concentric constraint forces selected circles, arcs, or ellipses to maintain the same center point.

Apply a Concentric Constraint

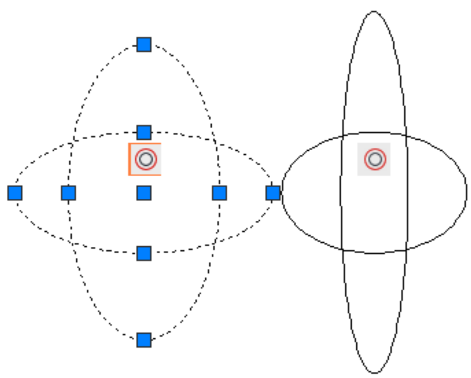
In the example below, a concentric constraint has been applied to the two circles on the left. On the right, the circles share the same center point.



Manipulate a Concentric Constraint

When you manipulate a block with a concentric constraint applied, the specified circles, arcs, or ellipses will always share the same center point.

In the example below, the constrained ellipses on the top have been manipulated to produce the block on the bottom. The two ellipses can move, change shape, or change size, but will always be constrained to share the same center point.



See also:

- Apply or Remove Geometric Constraints on page 725
- To apply a concentric constraint on page 729

Quick Reference

Commands

GEOMCONSTRAINT

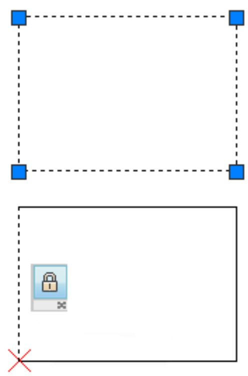
Applies or persists geometric relationships between objects or points on objects.

Add a Fixed Constraint

A fixed constraint fixes a point or curve to a specified location and orientation relative to the World Coordinate System (WCS).

Apply a Fixed Constraint

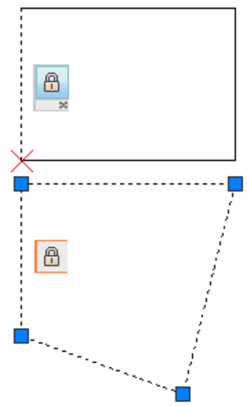
In the example below, a fix constraint has been applied to the rectangle on the top. The result is the block definition on the bottom, where the bottom-left corner of the rectangle is fixed to the specified coordinate. The red X indicates the constrained point on the rectangle.



Manipulate a Fixed Constraint

When you manipulate a block with a fix constraint applied, the specified point will always remain positioned at the same WCS coordinates.

In the example below, the geometry of the rectangle on the top has been manipulated. The other three corners of the rectangle can move, but the constrained point will remain in the same position.



See also:

- Apply or Remove Geometric Constraints on page 725
- To apply a fix constraint on page 729

Quick Reference

Commands

GEOMCONSTRAINT

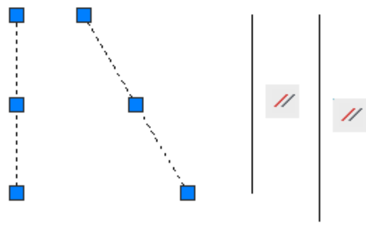
Applies or persists geometric relationships between objects or points on objects.

Add a Parallel Constraint

A parallel constraint forces two lines to remain parallel to each other.

Apply a Parallel Constraint

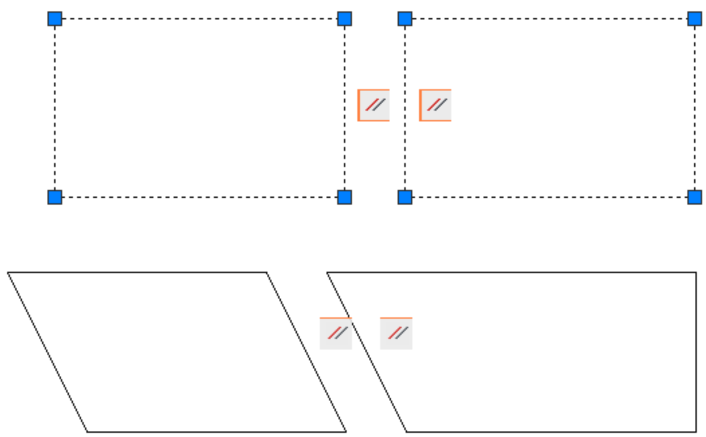
In the example below, a parallel constraint has been applied to the two lines on the left. The result is the block definition on the right, where the lines are parallel to each other.



Manipulate a Parallel Constraint

When you manipulate a block with a parallel constraint applied, the specified lines will always remain parallel to each other.

In the example below, a parallel constraint has been applied to one line in each of the rectangles on top. The geometry of each rectangle on the bottom has been changed, but the constrained lines remain parallel to each other.



See also:

- Apply or Remove Geometric Constraints on page 725
- To apply a parallel constraint on page 729

Quick Reference

Commands

GEOMCONSTRAINT

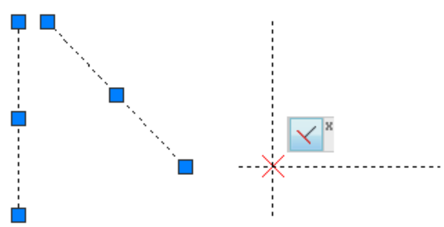
Applies or persists geometric relationships between objects or points on objects.

Add a Perpendicular Constraint

A perpendicular constraint forces two lines or polyline segments to maintain a 90-degree angle to each other.

Apply a Perpendicular Constraint

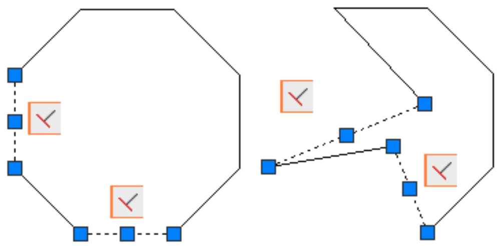
In the example below, a perpendicular constraint has been applied to the two lines on the left. The result is the block definition on the right, where the lines are perpendicular to each other.



Manipulate a Perpendicular Constraint

When you manipulate a block with a perpendicular constraint applied, the specified lines will always remain perpendicular to each other.

In the example below, a perpendicular constraint has been applied to one point on the octagon on the left. The geometry of the octagon on the right has been changed, but the constrained lines remain perpendicular to each other.



See also:

- Apply or Remove Geometric Constraints on page 725
- To apply a perpendicular constraint on page 730

Quick Reference

Commands

GEOMCONSTRAINT

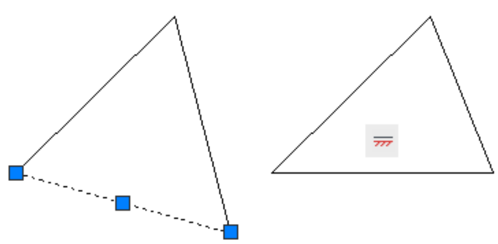
Applies or persists geometric relationships between objects or points on objects.

Add a Horizontal Constraint

A horizontal constraint forces a line or pair of points to remain parallel to the X-axis of the current UCS.

Apply a Horizontal Constraint

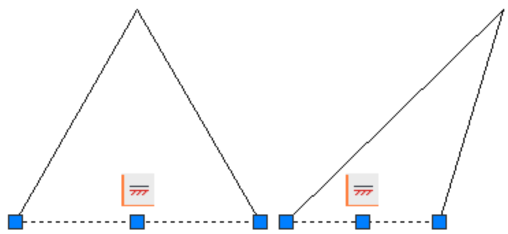
In the example below, a horizontal constraint has been applied to the triangle on the left. The result is the block definition on the right, where the constrained line is horizontal to the current UCS.



Manipulate a Horizontal Constraint

When you manipulate a block with a horizontal constraint applied, the constrained line or points will always remain horizontal to the UCS.

In the example below, a horizontal constraint has been applied to the triangle on the left. The geometry of the triangle on the right has been changed, but the constrained line remains horizontal.



See also:

- Apply or Remove Geometric Constraints on page 725
- To apply a horizontal constraint on page 730

Quick Reference

Commands

GEOMCONSTRAINT

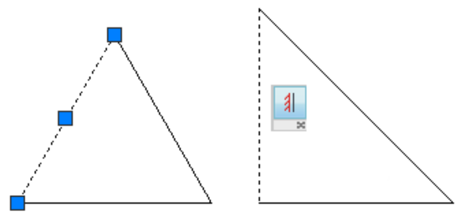
Applies or persists geometric relationships between objects or points on objects.

Add a Vertical Constraint

A vertical constraint forces lines of pairs of points to remain parallel to the Y-axis of the current UCS.

Apply a Vertical Constraint

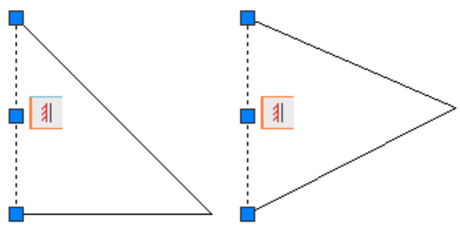
In the example below, a vertical constraint has been applied to the triangle on the left. The result is the block definition on the right, where the constrained line is vertical to the current UCS.



Manipulate a Vertical Constraint

When you manipulate a block with a vertical constraint applied, the constrained line or points will always remain vertical to the UCS.

In the example below, a vertical constraint has been applied to the triangle on the left. The geometry of the triangle on the right has been changed, but the constrained line remains vertical.



See also:

- Apply or Remove Geometric Constraints on page 725
- To apply a vertical constraint on page 730

Quick Reference

Commands

GEOMCONSTRAINT

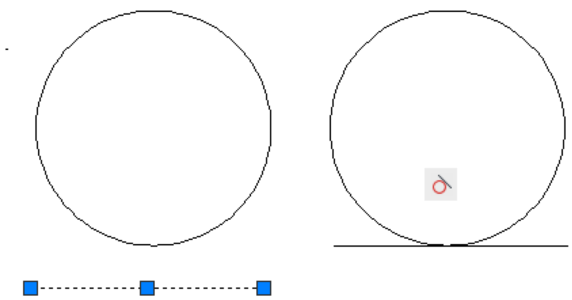
Applies or persists geometric relationships between objects or points on objects.

Add a Tangent Constraint

A tangent constraint forces two curves to maintain a point of tangency to each other or their extensions.

Apply a Tangent Constraint

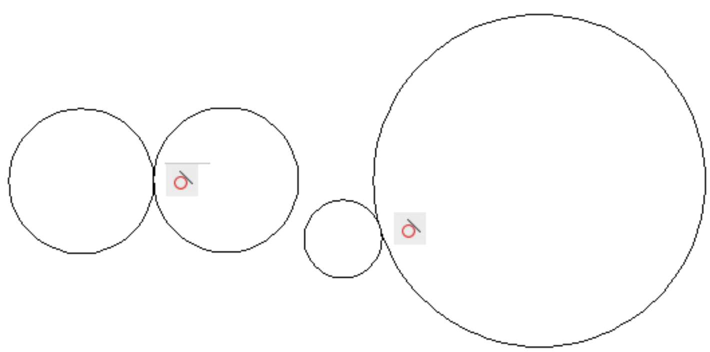
In the example below, a tangent constraint has been applied to the circle and line on the left. The result is the block definition on the right, where the constrained line is tangent to the circle.



Manipulate a Tangent Constraint

When you manipulate a block with a tangent constraint applied, the object selected second will always remain tangent to the object selected first.

In the example below, a tangent constraint has been applied to the two circles on the left. The size and rotation of the circles on the right have been changed, but the tangency remains.



See also:

- Apply or Remove Geometric Constraints on page 725

- To apply a tangent constraint on page 731

Quick Reference

Commands

GEOMCONSTRAINT

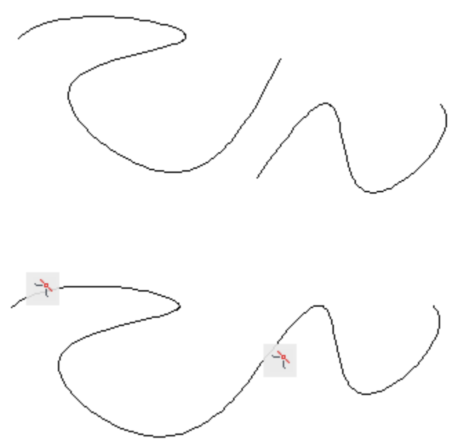
Applies or persists geometric relationships between objects or points on objects.

Add a Smooth Constraint

A smooth constraint forces a spline to maintain geometric continuity with another spline, line, arc, or polyline.

Apply a Smooth Constraint

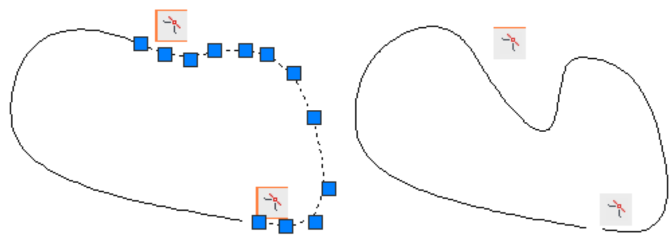
In the example below, a smooth constraint has been applied to the two spline curves on the left. The result is the block definition on the right, where the spline curves are joined continuously.



Manipulate a Smooth Constraint

When you manipulate a block with a smooth constraint applied, the constrained objects will always maintain fluid continuity.

In the example below, the two spline curves on the left have a smooth constraint applied. The geometry of the spline curves on the right has been changed, but they retain a fluid continuity.



See also:

- Apply or Remove Geometric Constraints on page 725
- To apply a smooth constraint on page 731

Quick Reference

Commands

GEOMCONSTRAINT

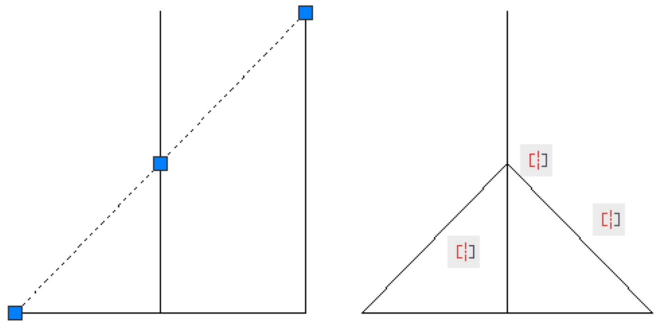
Applies or persists geometric relationships between objects or points on objects.

Add a Symmetric Constraint

A symmetric constraint forces two curves or points on an object to maintain symmetry about a selected line.

Apply a Symmetric Constraint

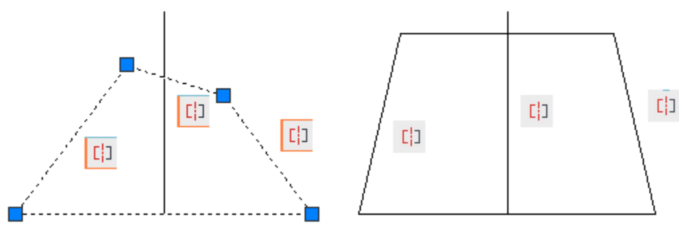
In the example below, a symmetric constraint has been applied to the two lines on the left. The result is the block definition on the right, where the lines are parallel to each other.



Manipulate a Symmetric Constraint

When you manipulate a block with a symmetric constraint applied, the constrained line's angle, or constrained circle's center point and radius, will remain symmetrical.

In the example below, the two outside edges of the polygon on the left have a symmetric constraint applied. The geometry of the polygon on the right has been changed, but the angles of the constrained edges remain symmetrical.



See also:

- Apply or Remove Geometric Constraints on page 725
- To apply a symmetric constraint on page 731
- Create Construction Geometry Within a Block on page 1002

Quick Reference

Commands

GEOMCONSTRAINT

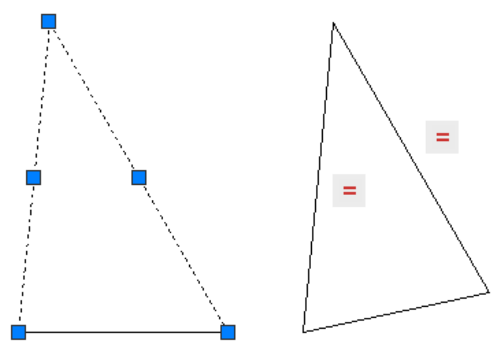
Applies or persists geometric relationships between objects or points on objects.

Add an Equal Constraint

An equal constraint forces two lines or polyline segments to maintain equal lengths or arcs.

Apply an Equal Constraint

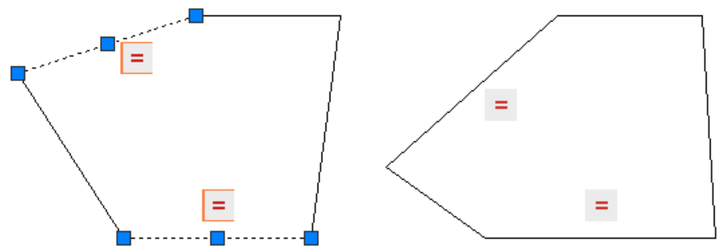
In the example below, an equal constraint has been applied to the triangle on the left. The result is the block definition on the right, where the constrained lines in the triangle are equal to each other.



Manipulate an Equal Constraint

When you manipulate a block with an equal constraint applied, the constrained objects will remain equal.

In the example below, two lines in the polygon on the left have an equal constraint applied. The geometry of the polygon on the right has been changed, but the constrained lines remain equal in length.



See also:

- Apply or Remove Geometric Constraints on page 725
- To apply an equal constraint on page 732

Quick Reference

Commands

GEOMCONSTRAINT

Applies or persists geometric relationships between objects or points on objects.

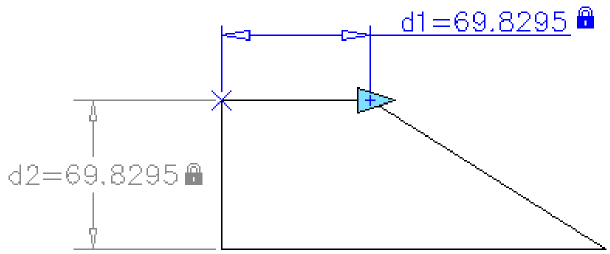
Add Dimensional Constraints to Dynamic Blocks

A dimensional constraint applied within the Block Editor using the BCPARAMETER command is called a **constraint parameter**.

Overview of Constraint Parameters

While you can use both dimensional constraints and constraint parameters in a block definition, only constraint parameters will display editable custom properties for that block reference.

Constraint parameters contain parameter information. The parameter values can be displayed or edited for the block reference. A constraint parameter can only be created in the Block Editor.

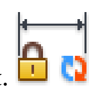


The above block definition contains both a linear constraint and a horizontal constraint parameter. The horizontal constraint parameter includes a grip, while the linear constraint does not. The horizontal constraint parameter is also dynamic, while the linear constraint is not.

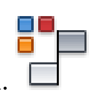
See also:

- Apply Dimensional Constraints on page 744
- Specify Custom Properties for Dynamic Blocks on page 1004
- Associative Dimensions on page 1627

To convert a dimensional constraint into a constraint parameter

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. Click Block Editor tab ➤ Dimensional panel ➤ Convert. 
3. Select a dimensional constraint in the Block Editor or enter a constraint parameter type option in the command line.

To turn on or turn off the constraint display status

- In the Block Editor status bar, click the Constraint Status button. 

To display or hide dynamic constraints

➤ Click Parametric tab ➤ Dimensional panel ➤ Dynamic Constraints.
DYNCONSTRAINTDISPLAY

Quick Reference

Commands

BCPARAMETER

Applies constraint parameters to selected objects, or converts dimensional constraints to parameter constraints.

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects.

System Variables

BCONSTATUSMODE

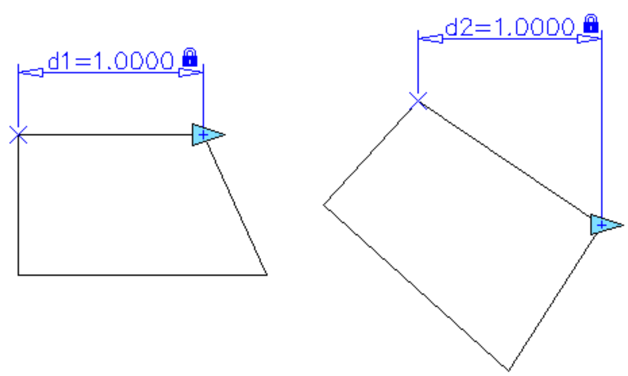
Turns the constraint display status on and off and controls the shading of objects based on their constraint level.

Add a Horizontal Constraint Parameter

A horizontal constraint parameter controls the X distance between two points on one object, or between two objects.

While a linear dimension on page 1679 represents the distance between two points, a *horizontal constraint parameter* defines and controls the horizontal distance between two points.

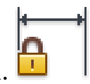
In the example below, the polygon on the left has a horizontal constraint of 1 unit applied to the top line of the polygon. When that polygon is rotated, the result is the polygon on the right. The constrained points will remain 1 unit apart horizontally.



See also:

- Apply Dimensional Constraints on page 744
- Constrain a Design with Formulas and Equations on page 758
- Create Horizontal and Vertical Dimensions on page 1679

To add a horizontal constraint to a block definition

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. Click Block Editor tab ➤ Dimensional panel ➤ Horizontal. 
3. Specify the first constraint point or object.
4. Specify the second constraint point or object.
5. Specify the dimension line location.
6. Follow the Command prompts to specify the following for the constraint:
   - Value or name and value
   - Number of grips
7. Press Enter.

Command entry: BCPARAMETER

Quick Reference

Commands

BCPARAMETER

Applies constraint parameters to selected objects, or converts dimensional constraints to parameter constraints.

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects.

System Variables

BCONSTATUSMODE

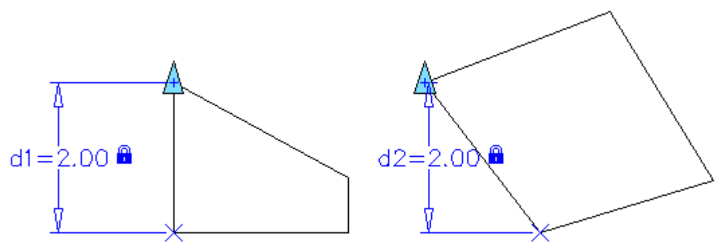
Turns the constraint display status on and off and controls the shading of objects based on their constraint level.

Add a Vertical Constraint Parameter

A vertical constraint controls the Y distance between two points on one object, or between two objects.

While a linear dimension on page 1679 represents the distance between two points, a *vertical constraint parameter* defines and controls the vertical distance between two points.

In the example below, the polygon on the left has a vertical constraint of 2 units applied to the left line of the polygon. When that polygon is rotated counterclockwise, the result is the polygon on the right. The constrained points will remain 2 units apart vertically.



See also:

- Apply Dimensional Constraints on page 744
- Constrain a Design with Formulas and Equations on page 758
- Create Horizontal and Vertical Dimensions on page 1679

To add a vertical constraint parameter to a block definition

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 Click Block Editor tab ➤ Dimensional panel ➤ Vertical. 

3 Specify the first constraint point or object.

4 Specify the second constraint point or object.

5 Specify the dimension line location.

6 Follow the Command prompts to specify the following for the constraint:
   - Value or name and value
   - Number of grips

7 Press Enter.

Command entry: BCPARAMETER

Quick Reference

Commands

BCPARAMETER

Applies constraint parameters to selected objects, or converts dimensional constraints to parameter constraints.

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects.

System Variables

BCONSTATUSMODE

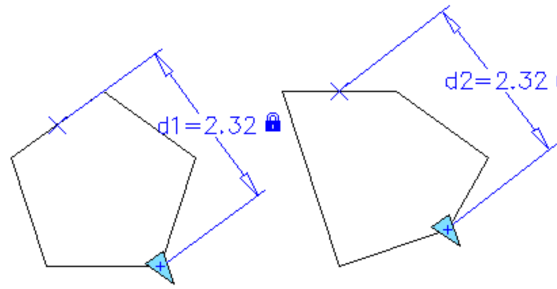
Turns the constraint display status on and off and controls the shading of objects based on their constraint level.

Add an Aligned Constraint Parameter

An aligned constraint controls the distance between two points on an object, one point and one object, or two line segments.

While analigned dimension on page 1681 represents the distance between two points, an *aligned constraint parameter* defines and controls the distance between two points.

In the example below, two points on the polygon on the left have an aligned constraint parameter applied. When that polygon is manipulated, the result is the polygon on the right. The constrained points will remain at the same distance, and the other points will move to keep that distance.



See also:

- Apply Dimensional Constraints on page 744
- Constrain a Design with Formulas and Equations on page 758
- Create Aligned Dimensions on page 1681

To add an aligned constraint to a block definition

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. Click Block Editor tab ➤ Dimensional panel ➤ Aligned. 
3. Specify the first constraint point or object.
4. Specify the second constraint point or object.
5. Specify the dimension line location.
6. Follow the Command prompts to specify the following for the constraint:
   - Value or name and value
   - Number of grips
7. Press Enter.

Command entry: BCPARAMETER

Quick Reference

Commands

BCPARAMETER

Applies constraint parameters to selected objects, or converts dimensional constraints to parameter constraints.

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects.

System Variables

BCONSTATUSMODE

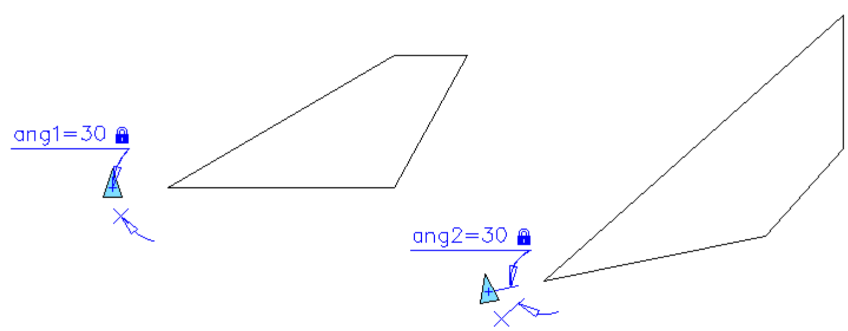
Turns the constraint display status on and off and controls the shading of objects based on their constraint level.

Add an Angular Constraint Parameter

An angular constraint controls the angle between two line or polyline segments, or the angle of an arc.

While an angular dimension on page 1696 represents the angle between two lines or three points, an *angular constraint parameter* defines and controls the angle.

In the example below, the polygon on the left has an angular constraint of 30 degrees applied to one corner of the polygon. When one point on that polygon is manipulated, the result is the polygon on the right. The constrained corner will remain at 30 degrees, while the geometry of the polygon will change to keep that angle.



See also:

- Apply Dimensional Constraints on page 744
- Create Angular Dimensions on page 1696

To add an angular constraint to a block definition

1. Click Insert tab ➤ Block panel ➤ Block Editor.
2. Click Block Editor tab ➤ Dimensional panel ➤ Angular.
3. Specify a line or a polyline segment.
4. Specify the second line or the next two constraint points (for polyline segments).
5. Specify the dimension line location.
6. Follow the Command prompts to specify the following for the constraint:
   - Value or name and value
   - Number of grips
7. Press Enter.

Command entry: BCPARAMETER

To add an angular constraint to an arc block definition

1. Click Insert tab ➤ Block panel ➤ Block Editor.
2. Click Block Editor tab ➤ Dimensional panel ➤ Angular.
3. Specify an arc.
4. Specify the dimension line location.
5. Follow the Command prompts to specify the following for the constraint:
   - Value or name and value

- Number of grips

6 Press Enter.

Command entry: BCPARAMETER

Quick Reference

Commands

BCPARAMETER

Applies constraint parameters to selected objects, or converts dimensional constraints to parameter constraints.

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects.

System Variables

BCONSTATUSMODE

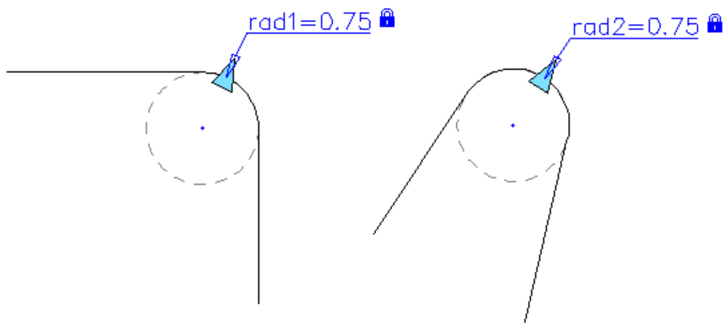
Turns the constraint display status on and off and controls the shading of objects based on their constraint level.

Add a Radial Constraint Parameter

A radial constraint parameter controls the radius of a circle, arc, or polyline arc segment.

While a radial dimension on page 1688 represents the radius of a circle or arc,, a *radial constraint parameter* defines and controls the radius.

In the example below, a radial constraint parameter has been applied to the polyline arc in the block definition on the left. The polyline has been manipulated in the block definition on the right. As a result, the radius of the arc remains fixed while the polyline segments around it change.



See also:

- Apply Dimensional Constraints on page 744
- Constrain a Design with Formulas and Equations on page 758
- Create Radial Dimensions on page 1688

To add a radial constraint to a block definition

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. Click Block Editor tab ➤ Dimensional panel ➤ Radius. 
3. Specify an arc or a circle.
4. Specify the dimension line location.
5. Follow the Command prompts to specify the following for the constraint:
   - Value or name and value
   - Number of grips
6. Press Enter.

Command entry: BCPARAMETER

Quick Reference

Commands

BCPARAMETER

Applies constraint parameters to selected objects, or converts dimensional constraints to parameter constraints.

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects.

System Variables

BCONSTATUSMODE

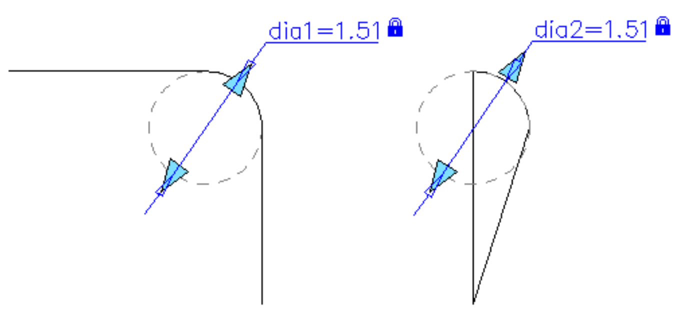
Turns the constraint display status on and off and controls the shading of objects based on their constraint level.

Add a Diameter Constraint Parameter

A diameter constraint parameter controls the radius of a circle, arc, or polyline arc segment.

While a diameter dimension on page 1688 represents the diameter of an arc or circle, a *diameter constraint parameter* defines and controls the diameter.

In the example below, a diameter constraint parameter has been applied to the polyline arc in the block definition on the left. The polyline has been manipulated in the block definition on the right. As a result, the diameter of the arc remains fixed while the polyline segments around it change.



See also:

- Apply Dimensional Constraints on page 744

- [Constrain a Design with Formulas and Equations](#) on page 758
- [Create Radial Dimensions](#) on page 1688

To add a diameter constraint to a block definition

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 Click Block Editor tab ➤ Dimensional panel ➤ Diameter. 

3 Specify an arc or a circle.

4 Specify the dimension line location.

5 Follow the Command prompts to specify the following for the constraint:
   - Value or name and value
   - Number of grips

6 Press Enter.

Command entry: BCPARAMETER

Quick Reference

Commands

BCPARAMETER

Applies constraint parameters to selected objects, or converts dimensional constraints to parameter constraints.

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects.

System Variables

BCONSTATUSMODE

Turns the constraint display status on and off and controls the shading of objects based on their constraint level.

Identify Fully Constrained Objects

When an object is fully constrained, all relevant geometric and dimensional constraints are applied to the geometry.

You should always fully constrain geometry in a dynamic block definition that contains constraints. If a block definition is not fully constrained, you may get unpredictable behavior when the block is inserted in a drawing.

For more information on fully constrained objects, see Overview of Constraints on page 718.

Work with Fully Constrained Block Definitions

You can identify fully constrained objects with the constrained status mode in the Block Editor. The BCONSTATUSMODE system variable allows you to switch between the constrained status modes. You can also use the ribbon to switch between modes while in the Block Editor.

The constrained geometry is represented with different colors. For more information see the Block Editor Settings dialog box.

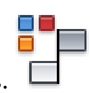
NOTE A fully constrained block definition must contain at least one fixed geometric constraint.

See also:

- Add a Fixed Constraint on page 904

To identify the constrained objects

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 Click Block Editor tab ➤ Manage panel ➤ Constraint Status. 

The constrained objects are represented with the constraint status colors specified in the Block Editor Settings dialog box.

Pointing device: In the Drawing Status Bar, click the Constraint Status button.

To specify color assignment for the constraint display status

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 Click Block Editor tab ➤ Manage panel ➤ Block Editor Settings.

3 In the Block Editor Settings dialog box, under Constraint Status, specify colors for the following constrained objects:
   - Unconstrained
   - Partially Constrained
   - Fully Constrained
   - Improperly Constrained

4 Click OK.

Command entry: BESETTINGS
Pointing device: In the Drawing Status Bar, right-click the Constraint Status button and click Settings.

Quick Reference

Commands

BESETTINGS

Displays the Block Editor Settings dialog box.

System Variables

BCONSTATUSMODE

Turns the constraint display status on and off and controls the shading of objects based on their constraint level.

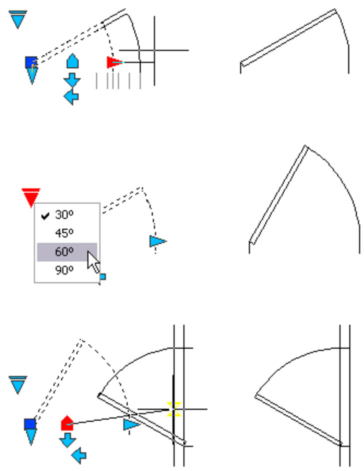
Add Actions and Parameters to Dynamic Blocks

You can add actions and parameters to dynamic blocks to determine how a block should behave when it is inserted into a drawing.

Overview of Actions and Parameters in Dynamic Blocks

While you work, you can manipulate the geometry in a dynamic block reference through custom grips or custom properties instead of searching for another block to insert, or redefining the existing one.

For example, the size of a door block reference in a drawing might need to change while you're editing the drawing. If the block is dynamic and defined to have an adjustable size, you can drag the custom grip or by specify a different size in the Properties palette. The door block might also contain an alignment grip, which allows you to align the door block reference easily to other geometry in the drawing.



You can create a block from scratch, or you can add dynamic behavior to an existing block definition. You can also create geometry, just as you would in the drawing area.

Parameters and actions are displayed only in the Block Editor. When you insert a dynamic block reference in a drawing, the parameters and actions contained in the dynamic block definition are not displayed.

Parameter Type	**Grip Type**	**Actions You Can Associate with a Parameter**
Point	 Standard	Move, Stretch
Linear	 Linear	Move, Scale, Stretch, Array

Parameter Type	Grip Type	Actions You Can Associate with a Parameter
Polar	Standard	Move, Scale, Stretch, Polar Stretch, Array,
XY	Standard	Move, Scale, Stretch, Array
Rotation	Rotation	Rotate
Flip	Flip	Flip
Alignment	Alignment	None (The action is implied and contained within the parameter.)
Visibility	Lookup	None (The action is implied and controlled by visibility states.)
Lookup	Lookup	Lookup
Base	Standard	None

See also:

■ Save Blocks on page 896

To create a dynamic block

1 Click Insert tab ➤ Block panel ➤ Block Editor.

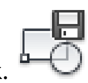
2 In the Edit Block Definition dialog box, do one of the following:
   ■ Select a block definition from the list.

- Select <Current Drawing> if you want to save the drawing as a dynamic block.
- Under Block to Create or Edit, enter a name for a new block definition.

3 Click OK.

4 In the Block Editor, add or edit geometry as necessary.

5 Do one of the following:
   - Add one or more parameter sets from the Parameter Sets tab of the Block Authoring Palettes, following the Command prompts. 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.
   - Add one or more parameters from the Parameters tab of the Block Authoring Palettes, following the Command prompts. Add one or more actions from the Actions tab, following the Command prompts.

6 Click Block Editor tab ➤ Open/Save panel ➤ Save Block. 

7 Click Close Block Editor.

Toolbar: Standard 
Command entry: BEDIT
Shortcut menu: Right-click a block. Click Block Editor.

Quick Reference

Commands

BEDIT

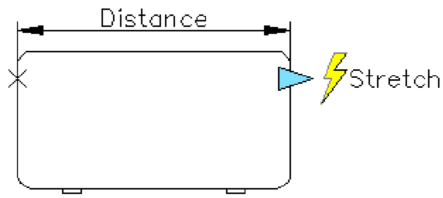
Opens the block definition in the Block Editor.

BLOCK

Creates a block definition from selected objects.

Quick Start to Creating Dynamic Blocks

You add dynamic behavior to new or existing block definitions by adding parameters and actions to the block in the Block Editor. In the following example, a desk block is shown in the Block Editor. The block contains a linear parameter, which displays similar to a dimension and is labeled "Distance," and a stretch action, which displays a lightning bolt and a "Stretch" label.



For a block to be dynamic, you must add at least one parameter. You then add an action and associate the action with the parameter. The types of parameters and actions you add to the block definition define how the block reference will work in a drawing.

Process for Creating Dynamic Blocks

To create quality dynamic blocks, so you get the results you expect, it is recommended that you follow the steps in the following process. This process will help you author dynamic blocks effectively.

Step 1. Plan the contents of the dynamic block before you create it

Before you create a dynamic block, you should know what it will look like and how it will be used in a drawing. Decide which objects within the block will change or move when the dynamic block reference is manipulated. Furthermore, decide *how* these objects will change. For example, you might create a dynamic block that can be resized. In addition, when the block reference is resized, additional geometry may be displayed. These factors determine the types of parameters and actions you add to the block definition, and how you make the parameters, actions, and geometry work together.

Step 2. Draw the geometry

You can draw the geometry for your dynamic block in the drawing area, the Block Editor contextual tab, or the Block Editor. You can also use existing geometry in a drawing or an existing block definition.

NOTE If you will use visibility states to change how geometry is displayed in the dynamic block references, you may not want to include all the geometry at this point. For more information about working with visibility states, see Create Visibility States.

Step 3. Understand how the block elements will work together

Before you add parameters and actions to your block definition, understand their *dependencies* on each other and on the geometry within the block. When you add an action to the block definition, you will need to associate the action with a parameter and a selection set of geometry. This creates a dependency. When you add multiple parameters and actions to a dynamic block reference, you will need to set up the correct dependencies in order for the block reference to function properly in a drawing.

For example, you create a dynamic block that contains several objects. Some of the objects have a stretch action associated with them. You also want all the objects to rotate around the same base point. In this case, you should add the rotate action after adding all the other parameters and actions. If the rotate action isn't associated with all of the other objects (geometry, parameters, and actions) in the block definition, parts of the block reference may not rotate or manipulating the block reference may cause unexpected results.

Step 4. Add parameters

Add the appropriate parameters to the dynamic block definition, following the prompts at the Command prompt. For more information about using parameters, see Add Action Parameters to Dynamic Blocks on page 936.

NOTE You can use the Parameter Sets tab of the Block Authoring Palettes to add a parameter and an associated action at the same time. For more information about using parameter sets, see Use Parameter Sets on page 984.

Step 5. Add actions

Add the appropriate actions to the dynamic block definition. Follow the prompts at the Command prompt, making sure to associate the actions with the correct parameters and geometry. For more information about using actions, see Overview of Using Actions on page 960.

Step 6. Define how the dynamic block reference will be manipulated

You can specify how the dynamic block reference will be manipulated in the drawing. You can manipulate a dynamic block reference through custom grips and custom properties. When you create a dynamic block definition, you define what grips are displayed and how they edit the dynamic block reference. You also specify whether or not custom properties for the block will be shown in the Properties palette and whether or not these properties can be changed through the palette or through custom grips.

Step 7. Test the block

On the ribbon, in the Block Editor contextual tab, Open/Save panel, click Test Block to test the block before you save it.

See also:

- Overview of Actions and Parameters in Dynamic Blocks on page 931
- Overview of Dynamic Block Elements

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BLOCK

Creates a block definition from selected objects.

BTESTBLOCK

Displays a window within the Block Editor to test a dynamic block.

Add Action Parameters to Dynamic Blocks

Overview of Using Action Parameters

You can use parameters in block references to specify

- Positions
- Distances
- Angles

When you add a parameter to a block definition, custom grips and properties are automatically added to the block. You use these custom grips and properties to manipulate the block reference in the drawing.

When you add a parameter to a dynamic block definition, grips are added to key points of the parameter. *Key points* are the parts of a parameter that you use to manipulate the block reference. For example, a linear parameter has key points at its base point and end point. You can manipulate the parameter distance from either key point.

The type of parameter that you add to a dynamic block determines the type of grips that are added. Each type of parameter supports only certain types of actions.

For example, a block of a chair can contain an *Angle* property that defines a **rotation parameter**. The rotation parameter defines the axis on which the chair can be rotated as you edit.

If you add a point parameter to a dynamic block definition, the **point parameter** defines two custom properties for the block reference: *Position X* and *Position Y* (relative to the base point of the block reference.)

A dynamic block definition must contain at least one parameter. When a parameter is added to a dynamic block definition, grips associated with key points of the parameter are automatically added. You must then add an action to the block definition and associate the action with a parameter.

In a drawing, you use a grip or custom property in the Properties palette to manipulate the block reference. When you manipulate the block reference in a drawing, by moving a grip or changing the value of a custom property in the Properties palette, you change the value of the parameter that defines that custom property in the block. When you change the value of the parameter, it drives the action that is associated with that parameter, which changes the geometry (or a property) of the dynamic block reference.

Parameters also define and constrain values that affect the dynamic block reference's behavior in a drawing. Some parameters can have a fixed set of values, minimum and maximum values, or increment values. For example, a linear parameter used in a window block may have the following fixed set of values: 10, 20, 30, and 40. When the block reference is inserted in a drawing, you can only change the window to one of these values. Adding a value set to a parameter allows you to limit how the block reference is manipulated in a drawing. For more information about parameter value sets, see Specify Value Sets for Dynamic Blocks on page 1009.

You can also extract parameter values through the Attribute Extraction wizard or through an attribute extraction template file.

See also:

- Specify Value Sets for Dynamic Blocks on page 1009
- Overview of Specifying Custom Properties for Dynamic Blocks on page 1005
- Extract Data from Block Attributes on page 1034
- Extract Block Attribute Data (Advanced) on page 1035

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BPARAMETER

Adds a parameter with grips to a dynamic block definition.

BSAVE

Saves the current block definition.

Add a Point Parameter

A point parameter defines an X and Y location for the block in the drawing.

In the Block Editor, a point parameter displays similar to an ordinate dimension.

You can also associate the following actions with a point parameter:

- Move on page 962
- Stretch on page 968

If a grip is displayed for a point parameter, you can manipulate the parameter even if no action is associated with it.

The point parameter contains a property called Chain Actions. For more information about allowing chained actions for a parameter, see Allow Chained Actions for Dynamic Blocks on page 1014.

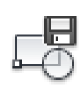
See also:

- Specify Grips for Dynamic Blocks on page 987
- Add a Move Action on page 962
- Add a Stretch Action on page 968

To add a point parameter to a dynamic block definition

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. Click Block Editor tab ➤ Action Parameters panel ➤ Point.
3. (Optional) Follow the Command prompts to specify the following for the parameter:
   - Type
   - Label
   - Description
   - Chained Actions
   - Properties displayed for block reference

   NOTE You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.

   ---
4. Specify a location for the parameter.
5. Specify a location for the parameter label.
6. To add an action now, double-click the alert icon. Follow the prompts to associate an action with the parameter and a selection set of geometry.
7. Click Block Editor tab ➤ Open/Save panel ➤ Save Block. 
8. If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard 
Command entry: BPARAMETER

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BPARAMETER

Adds a parameter with grips to a dynamic block definition.

BSAVE

Saves the current block definition.

Add a Linear Parameter

A linear parameter shows the distance between two anchor points.

When you insert a linear parameter, grip movement is constrained along a preset angle. In the Block Editor, a linear parameter looks similar to an aligned dimension.

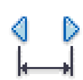
You can associate the following actions with a linear parameter:

- Move
- Scale
- Stretch
- Array

The linear parameter has a property called Chain Actions. For more information about allowing chained actions for a parameter, see Allow Chained Actions for Dynamic Blocks on page 1014.

To add a linear parameter to a dynamic block definition

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 Click Block Editor tab ➤ Dimensional panel ➤ Linear. 

3 (Optional) Follow the Command prompts to specify the following for the parameter:

- Type
- Label
- Description
- Value set
- Chained Actions
- Properties displayed for block reference

NOTE You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.

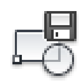
---

4 Specify a base point for the parameter.

5 Specify an endpoint for the parameter.

6 Specify a location for the parameter label

7 Follow the prompts to associate an action with the parameter and a selection set of geometry.

8 Click Block Editor tab ➤ Open/Save panel ➤ Save Block. 

9 If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard 
Command entry: BPARAMETER

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BPARAMETER

Adds a parameter with grips to a dynamic block definition.

BSAVE

Saves the current block definition.

Add a Polar Parameter

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 of the block reference. In the Block Editor, the polar parameter looks similar to an aligned dimension.

You can associate the following actions with a polar parameter:

- Move
- Scale
- Stretch
- Polar Stretch
- Array

The polar parameter has a property called Chain Actions. For more information about allowing chained actions for a parameter, see Allow Chained Actions for Dynamic Blocks on page 1014.

To add a polar parameter to a dynamic block definition

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

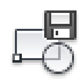
2 Click Block Editor tab ➤ Action Parameters panel ➤ Polar. 

3 (Optional) Follow the Command prompts to specify the following for the parameter:
   - Type
   - Label

- Description
- Value set
- Chained Actions
- Properties displayed for block reference

NOTE You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.

---

4. Specify a base point for the parameter.
5. Specify an endpoint for the parameter.
6. Specify a location for the parameter label
7. To add an action now, double-click the alert icon. Follow the prompts to associate an action with the parameter and a selection set of geometry.
8. Click Block Editor tab ➤ Open/Save panel ➤ Save Block. 
9. If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard 
Command entry: BPARAMETER

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BPARAMETER

Adds a parameter with grips to a dynamic block definition.

BSAVE

Saves the current block definition.

Add an XY Parameter

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).

You can associate the following actions with an XY parameter:

- Move
- Scale
- Stretch
- Array

The XY parameter has a property called Chain Actions. For more information about allowing chained actions for a parameter, see Allow Chained Actions for Dynamic Blocks on page 1014.

To add an XY parameter to a dynamic block definition

1. Click Insert tab ➤ Block panel ➤ Block Editor.
2. Click Block Editor tab ➤ Action Parameters panel ➤ XY.
3. Follow the Command prompts to specify the parameter type.
4. Specify a base point for the parameter.
5. Specify an endpoint for the parameter.
6. To add an action now, double-click the alert icon. Follow the prompts to associate an action with the parameter and a selection set of geometry.
7. Click Block Editor tab ➤ Open/Save panel ➤ Save Block.
8. If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard
Command entry: BPARAMETER

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BPARAMETER

Adds a parameter with grips to a dynamic block definition.

BSAVE

Saves the current block definition.

Add a Rotation Parameter

A rotation parameter defines an angle.

In the Block Editor, a rotation parameter displays as a circle.

You can associate a rotate action on page 975 with a rotation parameter.

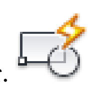
If a grip is displayed for a rotate parameter, you can manipulate the parameter even if no action is associated with it.

The rotation parameter has a property called Chain Actions. For more information about allowing chained actions for a parameter, see Allow Chained Actions for Dynamic Blocks on page 1014.

See also:

- Add a Rotate Action on page 975

To add a rotation parameter to a dynamic block definition

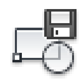
1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. Click Block Editor tab ➤ Action Parameters panel ➤ Rotation.
3. Follow the Command prompts to specify the parameter type.
4. Specify a base point for the parameter.
5. Specify the radius for the parameter.
6. Specify a base angle for the parameter.

The angle shown in the Properties palette when you select the block reference in a drawing is measured relative to the base angle specified in the block definition.

7 Specify a default rotation angle for the parameter.

8 Specify a location for the parameter label.

9 To add an action now, double-click the alert icon. Follow the prompts to associate an action with the parameter and a selection set of geometry.

10 Click Block Editor tab ➤ Open/Save panel ➤ Save Block. 

11 If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard 
Command entry: BPARAMETER

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BPARAMETER

Adds a parameter with grips to a dynamic block definition.

BSAVE

Saves the current block definition.

Add a Flip Parameter

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. The parameter displays a value that indicates whether the block reference has been flipped.

You can associate a flip action with a flip parameter.

See also:

- Add a Flip Action on page 978

To add a flip parameter to a dynamic block definition

1. Click Insert tab ➤ Block panel ➤ Block Editor.
2. Click Block Editor tab ➤ Action Parameter panel ➤ Flip.
3. (Optional) Follow the Command prompts to specify the following for the parameter:
   - Type
   - Label
   - Description
   - Properties displayed for block reference

   NOTE You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.

   ---

4. Specify the first point of the reflection line for the flip parameter.

   This first point of the reflection line is the default location for the flip parameter grip.
5. Specify the second point of the reflection line for the flip parameter.
6. Specify a location for the parameter label.
7. To add an action now, double-click the alert icon. Follow the prompts to associate an action with the parameter and a selection set of geometry.
8. Click Block Editor tab ➤ Open/Save panel ➤ Save Block.
9. If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard
Command entry: BPARAMETER

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BPARAMETER

Adds a parameter with grips to a dynamic block definition.

BSAVE

Saves the current block definition.

Add an Alignment Parameter

An alignment parameter defines an *X* and Y location and an angle.

An alignment parameter allows the block reference to automatically rotate around a point to align with another object in the drawing. An alignment parameter affects the rotation property of the block reference.

In the Block Editor, the parameter looks like an alignment line.

An alignment parameter always applies to the entire block and needs no action associated with it.

To add an alignment parameter to a dynamic block definition

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. Click Block Editor tab ➤ Action Parameter panel ➤ Alignment.
3. Follow the Command prompts to specify the parameter type.
4. Specify the base point of the alignment parameter.
   This base point is the default location for the alignment parameter grip.
5. (Optional) Enter **type** at the Command prompt and follow the prompts to specify whether the parameter type is perpendicular or tangent.
6. Specify the alignment direction.

7 Click Block Editor tab ➤ Open/Save panel ➤ Save Block.

8 If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard
Command entry: BPARAMETER

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BPARAMETER

Adds a parameter with grips to a dynamic block definition.

BSAVE

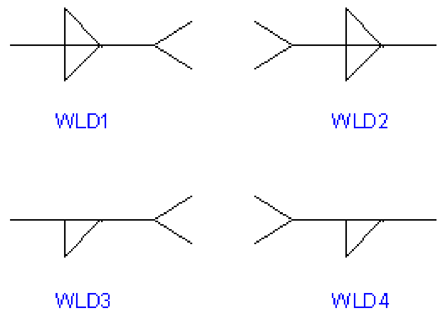
Saves the current block definition.

Add a Visibility Parameter

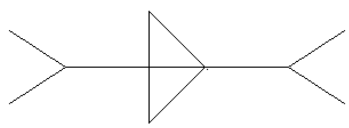
A visibility parameter controls the visibility of objects in the block.

You can create a block that has many different graphical representations. You can easily change a block reference that has different visibility states without having to find a different one to insert in your drawing.

For example, you have the following four different weld symbols.

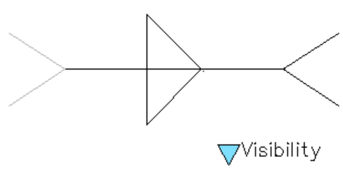


Using visibility states, you can combine these weld symbols into a single dynamic block. The following example shows the geometry for the four weld symbols combined in one dynamic block definition in the Block Editor.

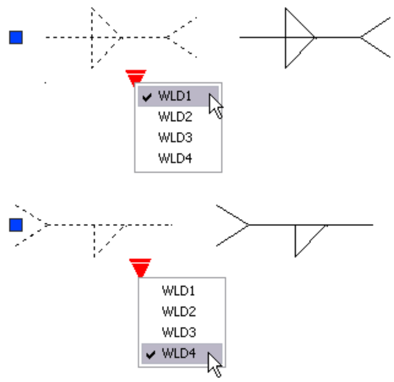


After you combine the geometry in the Block Editor, you add a visibility parameter. You add only one visibility parameter to a dynamic block definition. You do not associate any actions with a visibility parameter.

You can then create and name a different visibility state for each weld symbol (for example, WLD1, WLD2, WLD3, and WLD4). You can make particular geometry visible or invisible for each state. In the following example, the WLD1 visibility state is displayed in the Block Editor. The geometry that displays in a dimmed state is invisible for the WLD1 visibility state.



The visibility parameter includes a lookup grip. This grip is always displayed in a block reference that contains visibility states. When you click the grip in the block reference, a drop-down list of all the visibility states in the block reference is displayed. When you select one of the states from the list, the geometry that is visible for that state is displayed in the drawing.



The Block Editor contextual tab displays the name of the current visibility state. All blocks have at least one visibility state. You cannot delete the current state. This area of the tool bar also provides several tools for working with visibility states.

When you work with visibility states, you may or may not want to see the geometry that is invisible for a given state. You can use the Visibility Mode button (BVMODE) to display or not display geometry (in a dimmed state) that is invisible. When the BVMODE system variable is set to 1, geometry that is invisible for a given state displays in a dimmed mode.

To add a visibility parameter to a dynamic block definition

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 Click Block Editor tab ➤ Action Parameter panel ➤ Visibility.

3 (Optional) Follow the Command prompts to specify the following for the parameter:

- Type
- Label
- Description
- Properties displayed for block reference

NOTE You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.

4 Specify a location for the parameter.
This is the location for the visibility parameter grip in the block reference.

5 Click Block Editor tab ➤ Open/Save panel ➤ Save Block.

6 If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard
Command entry: BPARAMETER

To create a new visibility state based on the current state

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 Click Block Editor tab ➤ Visibility panel ➤ Visibility States.

3 In the Visibility States dialog box, click New.

4 In the New Visibility State dialog box, enter a name for the new visibility state.

5 Click Leave Visibility of Existing Objects Unchanged in New State.

6 Click OK.

The visibility of existing objects is unchanged in the new state.

Command entry: BVSTATE

To create a new visibility state in which all objects are invisible

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 Click Block Editor tab ➤ Visibility panel ➤ Visibility States.

3 In the Visibility States dialog box, click New.

4 In the New Visibility State dialog box, enter a name for the new visibility state.

5 Click Hide All Existing Objects in New State.

6 Click OK.

Command entry: BVSTATE

To create a new visibility state in which all objects are visible

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 Click Block Editor tab ➤ Visibility panel ➤ Visibility States. 

3 In the Visibility States dialog box, click New.

4 In the New Visibility State dialog box, enter a name for the new visibility state.

5 Click Show All Existing Objects in New State.

6 Click OK.

Command entry: BVSTATE

To make a visibility state current

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 Click Block Editor tab ➤ Visibility panel ➤ Visibility States. 

3 Select the visibility state that you want to set as the current state.

Command entry: BVSTATE
Shortcut menu: In the Visibility States dialog box, right-click a visibility state in the list. Click Set Current.

To delete a visibility state

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. Click Block Editor tab ➤ Visibility panel ➤ Visibility States. 
3. In the Visibility States dialog box, select a visibility state from the list.
4. Click Delete.
5. Click OK.

Command entry: BVSTATE
Shortcut menu: In the Visibility States dialog box, right-click a visibility state in the list. Click Delete State.

To rename a visibility state

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. Click Block Editor tab ➤ Visibility panel ➤ Visibility States. 
3. In the Visibility States dialog box, select a visibility state from the list.
4. Click Rename.
5. Enter a new name for the visibility state.
6. Click OK.

Command entry: BVSTATE
Shortcut menu: Right-click a visibility state in the list. Click Rename State.

To make objects visible in the current visibility state

1. Click Insert tab ➤ Block panel ➤ Block Editor. 

2 Click Block Editor tab ➤ Visibility panel ➤ Make Visible.

3 In the drawing area, select the objects to make visible in the current visibility state. Press Enter.

4 At the Command prompt, enter **current**. Press Enter.

Command entry: BVSHOW
Shortcut menu: In the Block Editor drawing area, select one or more objects. Right-click in the drawing area. Click Object Visibility ➤ Show For Current State.

To make objects visible in all visibility states

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 In the Block Editor drawing area, select the objects that you want to make visible in all visibility states.

3 Right-click in the drawing area.

4 Click Object Visibility ➤ Show For All States.

Command entry: BVSHOW

To make objects invisible in the current visibility state

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 Click Block Editor tab ➤ Visibility panel ➤ Make Invisible.

3 In the drawing area, select the objects that you want to make invisible in the current visibility state. Press Enter.

4 At the Command prompt, enter **current**. Press Enter.

Command entry: BVHIDE

Shortcut menu: In the Block Editor drawing area, select one or more objects. Right-click in the drawing area. Click Object Visibility ➤ Hide For Current State.

To make objects invisible in all visibility states

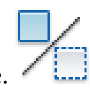
1 In the drawing area, select the objects that you want to make invisible in all visibility states.

2 Right-click in the drawing area.

3 Click Object Visibility ➤ Hide For All States.

Command entry: BVHIDE

To display or not display invisible geometry for visibility states

- Click Insert tab ➤ Block panel ➤ Block Editor. 
- Click Block Editor tab ➤ Visibility panel ➤ Visibility Mode. 

To change the order of the visibility states list for a dynamic block reference

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 Click Block Editor tab ➤ Visibility panel ➤ Visibility States. 

3 In the Visibility States dialog box, select a visibility state in the list.

4 Click Move Up or Move Down to change the order.

5 Repeat steps 2 and 3 until the list of visibility states is in the correct order.

6 Click OK.

Command entry: BVSTATE

To set the default visibility state for the dynamic block reference

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 Click Block Editor tab ➤ Visibility panel ➤ Visibility States.

3 In the Visibility States dialog box, select a visibility state in the list.

4 Click Move Up until the selected visibility state is at the top of the list.

5 Click OK.

Command entry: BVSTATE

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BPARAMETER

Adds a parameter with grips to a dynamic block definition.

BVHIDE

Makes objects invisible in the current visibility state or all visibility states in a dynamic block definition.

BVSHOW

Makes objects visible in the current visibility state or all visibility states in a dynamic block definition.

BVSTATE

Creates, sets, or deletes a visibility state in a dynamic block.

System Variables

BVMODE

Controls how objects that are made invisible for the current visibility state are displayed in the Block Editor.

Add a Lookup Parameter

A lookup parameter defines a custom property that you can specify or set to evaluate a value from a list or table you define.

A lookup parameter 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 with an associated grip.

You can associate a lookup parameter with a lookup action.

See also:

■ Add a Lookup Action on page 983

To add a lookup parameter to a dynamic block definition

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 Click Block Editor tab ➤ Action Parameter panel ➤ Lookup.

3 (Optional) Follow the Command prompts to specify the following for the parameter:

■ Type

■ Label

■ Description

■ Properties displayed for block reference

NOTE You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.

---

4 Specify a location for the parameter.

5 To add an action now, double-click the alert icon. Follow the prompts to associate a lookup action with the parameter. This will display the Property Lookup Table dialog box, which you can complete now or later.

6 Click OK.

7 Click Block Editor tab ➤ Open/Save panel ➤ Save Block.

8 If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard
Command entry: BPARAMETER

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BPARAMETER

Adds a parameter with grips to a dynamic block definition.

BSAVE

Saves the current block definition.

Add a Base Point Parameter

A base point parameter defines a base point for the dynamic block reference relative to the geometry in the block.

In the Block Editor, a base point parameter displays as a circle with crosshairs.

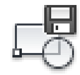
A base point parameter cannot be associated with any actions, but can belong to an action's selection set.

To add a base point parameter to a dynamic block definition

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 Click Block Editor tab ➤ Action Parameter panel ➤ Basepoint.

3 Specify a location for the parameter.

4 Click Block Editor tab ➤ Open/Save panel ➤ Save Block. 

5 If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard 
Command entry: BPARAMETER

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BPARAMETER

Adds a parameter with grips to a dynamic block definition.

BSAVE

Saves the current block definition.

Add Actions to Dynamic Blocks

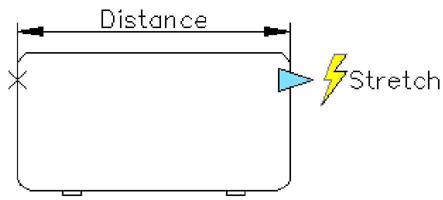
Actions define how the geometry of a dynamic block reference will move or change when the custom properties of the block reference are manipulated in a drawing.

Overview of Using Actions

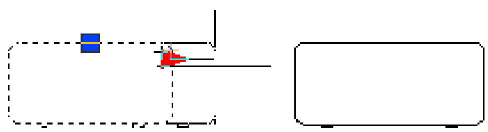
Actions define how the geometry of a dynamic block reference will move or change when the block reference is manipulated in a drawing.

In general, when you add an action to a dynamic block definition, you must associate the action with a parameter, a key point on the parameter, and geometry. A *key point* is the point on a parameter that drives its associated action when edited. The geometry associated with an action is called the *selection set*.

In the following example, the dynamic block definition contains geometry that represents a desk, a linear parameter with one grip specified for its endpoint, and a stretch action associated with the endpoint of the parameter and the geometry for the right side of the desk. The endpoint of the parameter is the key point. The geometry on the right side of the desk is the selection set.



When you want to change the block reference in a drawing, you move the grip, and the desk stretches.



You can assign more than one action to the same parameter and geometry.

NOTE Do not assign two or more of the same type of actions to the same key point on a parameter if both actions affect the same geometry. This can result in unexpected behavior in the block reference.

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BACTION

Adds an action to a dynamic block definition.

BACTIONSET

Specifies the selection set of objects associated with an action in a dynamic block definition.

BASSOCIATE

Associates an action with a parameter in a dynamic block definition.

Add a Move Action

The move action is similar to the MOVE command.

In a dynamic block definition, you can associate a move action with any of the following parameters:

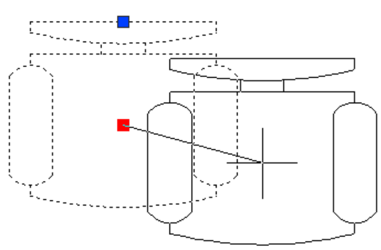
- Point on page 938
- Linear on page 940
- Polar on page 942
- XY on page 944

After associating a move action with a parameter, you associate the action with a selection set of geometry.

In a dynamic block reference, if you change the value of a parameter associated with a move action through a grip or the Properties palette, it may affect the key point on the parameter associated with the move action. If the key point is affected, the geometry in the move action's selection set will move.

For example, in a dynamic block that represents a chair, the block contains a point parameter and a move action associated with the point parameter. The move action's selection set contains all the geometry in the block. When you use the grip associated with the point parameter (or the Position X or Position Y properties in the Properties palette) to manipulate the dynamic block reference, it changes the value of the point parameter. This change in value causes the chair to move.

In the following example, when the chair block reference is moved by dragging the point parameter grip, the new position of this grip is reported in the Properties palette.



Specify the Distance Type Property for a Move Action

When a move action is associated with an XY parameter, the move action has an override property called *Distance Type*. This property specifies whether the distance applied to the move is the parameter's *X* value, *Y* value, or *X* and *Y* coordinate value from the parameter's base point.

For example, you specify X Distance as the Distance Type for a move action in a dynamic block definition. That means that the block can only be moved on the *X* axis. So if you try to move the block on the *Y* axis, it doesn't move.

See also:

- Add a Point Parameter on page 938
- Add a Linear Parameter on page 940
- Add a Polar Parameter on page 942
- Add an XY Parameter on page 944
- Use Distance Multiplier and Angle Offset Action Overrides on page 1017

To add a move action to a dynamic block definition

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. Click Block Editor tab ➤ Action Parameters panel ➤ Move.
3. In the Block Editor drawing area, select one of the following parameters to associate with the move action:
   - Point
   - Linear
   - Polar
   - XY
4. Select the objects (the selection set) to associate with the action. Press Enter.
5. (Optional) Follow the Command prompts to specify a distance multiplier and angle offset.
6. Specify the location of the action.

NOTE The action location in the block definition has no effect on the appearance or functionality of the block reference.

7 Click Block Editor tab ➤ Open/Save panel ➤ Save Block.

8 If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard
Command entry: BACTION

To specify the Distance Type property for a move action associated with an XY parameter

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 In the Block Editor, select a move action.

3 On the Properties palette, Overrides area, Distance Type, select an option from the list.

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BACTION

Adds an action to a dynamic block definition.

BACTIONSET

Specifies the selection set of objects associated with an action in a dynamic block definition.

BASSOCIATE

Associates an action with a parameter in a dynamic block definition.

PROPERTIES

Controls properties of existing objects.

Add a Scale Action

The scale action is similar to the SCALE command. In a dynamic block reference, a scale action causes the block's selection set to scale when the associated parameter is edited by moving grips or by using the Properties palette.

In a dynamic block definition, you associate a scale action with an entire parameter, not a key point on the parameter. You can associate a scale action with any of the following parameters:

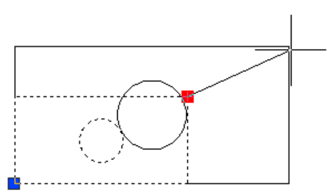
- Linear on page 940
- Polar on page 942
- XY on page 944

After associating a scale action with a parameter, you associate the action with a selection set of geometry.

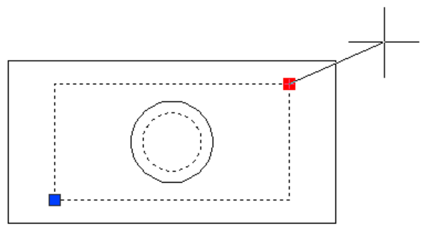
Specify the Type of Base Point for a Scale Action

The scale action has a property called *Base Type*. With this property, you specify whether the base point for the scale factor is *dependent* or *independent*.

If the base type is dependent, the objects in the selection set scale relative to the base point of the parameter with which the scale action is associated. In the following example, a scale action is associated with an XY parameter. The scale action base type is dependent. The base point of the XY parameter is located at the lower left corner of the rectangle. When the custom grip is used to scale the block, it scales relative to the lower-left corner of the rectangle.



If the base type is independent (shown in the Block Editor as an X marker), you specify a base point independent of the parameter with which the scale action is associated. The objects in the selection set will scale relative to this independent base point you specify. In the following example, a scale action is associated with an XY parameter. The scale action base type is independent. The independent base point is located at the center of the circle. When the custom grip is used to scale the block, it scales relative to the center of the circle.



Specify the Scale Type Property for a Scale Action

When a scale action is associated with an XY parameter, the scale action has an override property called *Scale Type*. This property specifies whether the scale factor applied is the parameter's *X* distance, *Y* distance, or *X* and *Y* coordinate value distance from the parameter's base point.

For example, you specify X Distance as the Scale Type for a scale action in a dynamic block definition. In a drawing, when you edit the block reference by dragging a grip on the XY parameter only along the Y axis, the associated geometry does not scale.

See also:

- Add a Linear Parameter on page 940
- Add a Polar Parameter on page 942
- Add an XY Parameter on page 944

To add a scale action to a dynamic block definition

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. Click Block Editor tab ➤ Action Parameters panel ➤ Scale.
3. In the Block Editor drawing area, select one of the following parameters to associate with the action:
   - Linear

- Polar
- XY

4 Select the geometry (the selection set) to associate with the action. Press Enter.

5 (Optional) Follow the Command prompts to specify the type of base point.

6 Specify the location of the action.

NOTE The action location in the block definition has no effect on the appearance or functionality of the block reference.

7 Click Block Editor tab ➤ Open/Save panel ➤ Save Block.

8 If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard
Command entry: BACTION

To specify an independent base point for a scale action in a dynamic block definition

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 In the Block Editor, select a scale action.

3 On the Properties palette, Overrides area, Base Type, select Independent from the drop-down list.

To specify the Scale Type property for a scale action associated with an XY parameter

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 In the Block Editor, select a scale action associated with an XY parameter.

3 On the Properties palette, Overrides area, Scale Type, select an option from the drop-down list.

Depending on the override you apply, the scale action for the block is limited to the *X* axis, the *Y* axis, or both.

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BACTION

Adds an action to a dynamic block definition.

BACTIONSET

Specifies the selection set of objects associated with an action in a dynamic block definition.

BASSOCIATE

Associates an action with a parameter in a dynamic block definition.

Add a Stretch Action

A stretch action causes objects to move and stretch a specified distance in a specified location.

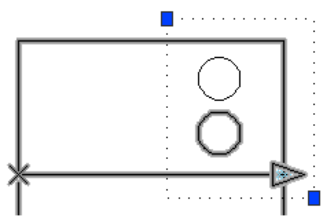
In a dynamic block definition, you can associate a stretch action with any of the following parameters:

- Point on page 938
- Linear on page 940
- Polar on page 942
- XY on page 944

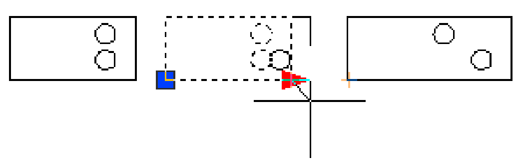
After associating a stretch action with a parameter, you specify a stretch frame for the stretch action. You then select the objects for the stretch action's selection set. The stretch frame determines how the objects within or crossed by the frame are edited in the block reference. The behavior is similar to specifying a crossing selection window with the STRETCH command.

- Objects entirely within the frame are moved.
- Objects that cross the frame are stretched.
- Objects within or crossed by the frame, but not included in the selection set, are not stretched or moved.
- Objects outside the frame and included in the selection set are moved.

In the following example, the stretch frame is indicated by the dashed line, and the selection set has a haloed effect. The top circle, while enclosed by the stretch frame, is not included in the selection set, so the top circle won't move. The bottom circle is entirely enclosed in the stretch frame and included in the selection set, so the bottom circle will move. The rectangle is crossed by the stretch frame and included in the selection set, so the rectangle will stretch.



In a dynamic block reference, if you change the value of a parameter associated with a stretch action through a grip or the Properties palette, it may affect the key point on the parameter associated with the stretch action. If the key point is affected, the geometry in the stretch action's selection set will move.



Specify the Distance Type Property for a Stretch Action

When a stretch action is associated with an XY parameter, the stretch action has an override property called *Distance Type*. This property specifies whether the distance applied to the move is the parameter's *X* value, *Y* value, or *X* an *Y* coordinate value from the parameter's base point.

For example, you specify X Distance as the Distance Type for a stretch action in a dynamic block definition. In a drawing, when you try to edit the block reference by dragging the key point only along the *Y* axis, the associated geometry does not move because you've added the Distance Type override allowing only a move on the *X* axis.

See also:

- Add a Point Parameter on page 938
- Add a Linear Parameter on page 940
- Add a Polar Parameter on page 942
- Add an XY Parameter on page 944
- Use Distance Multiplier and Angle Offset Action Overrides on page 1017

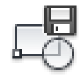
To add a stretch action to a block definition

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. Click Block Editor tab ➤ Action Parameters panel ➤ Stretch.
3. In the Block Editor drawing area, select one of the following parameters to associate with the action:
   - Point
   - Linear
   - Polar
   - XY

   NOTE If you associated the stretch action with a point parameter, then skip the next step.

4. Select a parameter point to associate with the action.
5. Specify the first corner of the stretch frame.
6. Specify the opposite corner of the stretch frame.
7. Select the objects for the selection set. Press Enter.
8. (Optional) Follow the Command prompts to specify a distance multiplier and angle offset.

9 Specify the location of the action.

10 Click Block Editor tab ➤ Open/Save panel ➤ Save Block. 

11 If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard 
Command entry: BACTION

To specify the Distance Type property for a stretch action associated with an XY parameter

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 In the Block Editor, select a stretch action associated with an XY parameter.

3 On the Properties palette, Overrides area, Distance Type, select an option from the drop-down list.

Depending on the override you apply, the stretch action for the block is limited to the *X* axis, the *Y* axis, or both.

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BACTION

Adds an action to a dynamic block definition.

BACTIONSET

Specifies the selection set of objects associated with an action in a dynamic block definition.

BASSOCIATE

Associates an action with a parameter in a dynamic block definition.

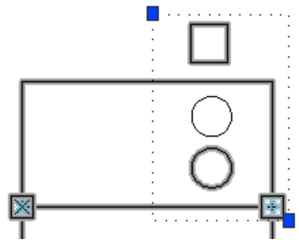
Add a Polar Stretch Action

In a dynamic block definition, you can only associate a polar stretch action with a polar parameter. The base point for the stretch part of the action is the parameter point opposite the key point.

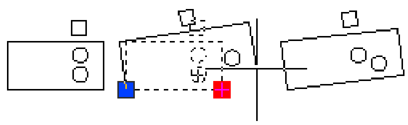
After associating a polar stretch action with a polar parameter on page 942, you specify a stretch frame for the polar stretch action. You then select objects to stretch and objects to rotate.

- Objects entirely within the frame are moved.
- Objects that cross the frame are stretched.
- Objects in the action's selection set specified to rotate only are not stretched.
- Objects within the frame are moved linearly after they are rotated.
- Objects crossed by the frame are stretched linearly after they are rotated.
- Objects within or crossed by the frame, but not included in the selection set, are not stretched or rotated.
- Objects outside the frame and included in the selection set are moved.

In the following example, the stretch frame is indicated by the dashed line, and the selection set has a haloed effect. The top circle, while enclosed by the stretch frame, is not included in the selection set, so the top circle won't move. The bottom circle is entirely enclosed in the stretch frame and included in the stretch selection set, so the bottom circle will move. The rectangle is crossed by the stretch frame and included in the selection set, so the rectangle will stretch. The square is entirely enclosed in the stretch frame and included in the rotate selection set, but not the stretch selection set, so the square will rotate only.



In a dynamic block reference, if you change the value of a parameter associated with a polar stretch action through a grip or the Properties palette, it may affect the key point on the parameter associated with the polar stretch action. If the key point is affected, the geometry in the polar stretch action's selection set will move or rotate depending on how the block was defined.



NOTE If you only want objects in the block reference to rotate, don't include any objects in the stretch frame.

See also:

- Add a Polar Parameter on page 942
- Use Distance Multiplier and Angle Offset Action Overrides on page 1017

To add a polar stretch action to a block definition

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. In the Block Editor, in the Block Authoring Palettes window, Actions tab, click the Stretch Action tool.
3. In the Block Editor drawing area, select a polar parameter to associate with the action. (You can associate only a polar parameter with a polar stretch action.)
4. Select a parameter point to associate with the action.
5. Specify the first corner of the stretch frame.

6 Specify the opposite corner of the stretch frame.

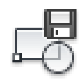
7 Select the objects to stretch or move. Press Enter.

8 Select the objects that you want to rotate. Press Enter.

9 (Optional) Follow the Command prompts to specify distance and offset.

10 Specify the location of the action.

> **NOTE** The action location in the block definition has no effect on the appearance or functionality of the block reference.

11 Click Block Editor tab ➤ Open/Save panel ➤ Save Block. 

12 If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard 
Command entry: BACTION

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BACTION

Adds an action to a dynamic block definition.

BACTIONSET

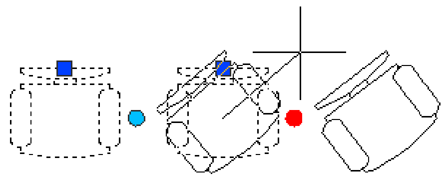
Specifies the selection set of objects associated with an action in a dynamic block definition.

BASSOCIATE

Associates an action with a parameter in a dynamic block definition.

Add a Rotate Action

The rotate action is similar to the ROTATE command.



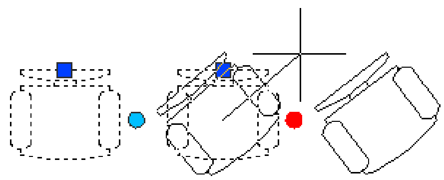
In a dynamic block definition, you can associate a rotate action only with a rotation parameter on page 945. The rotate action is associated with the entire parameter, not a key point on the parameter.

After associating a rotate action with a rotation parameter, you associate the action with a selection set of geometry.

Specify the Type of Base Point for a Rotate Action

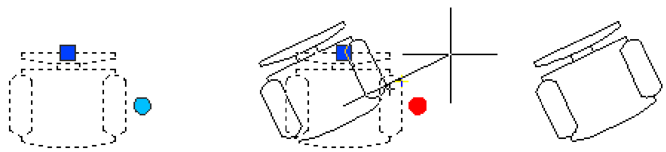
The rotate action has a property called *Base Type*. With this property, you specify whether the base point for the rotation is the base point of the parameter or an independent base point that you specify in the block definition.

By default, Base Type is set to Dependent. This means that the block rotates around the associated rotation parameter's base point. In the following example, the chair block contains a rotation parameter and an associated rotate action. The rotate action's Base Type is Dependent. The base point of the parameter is at the center of the chair. Thus, the chair rotates about the center point.

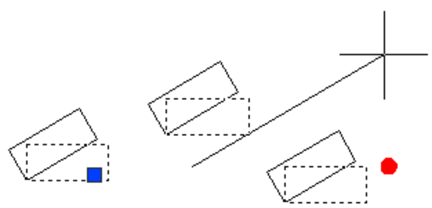


When you set Base Type to Independent, you can specify a base point for the rotate action other than the base point of the associated rotation parameter. This independent base point is shown in the Block Editor as an X marker. The location of the independent base point can be changed by dragging it or by editing the Base X and Base Y values in the Overrides section of the Properties palette.

In the following example, the chair block contains a rotation parameter and an associated rotate action. The rotate action's Base Type is Independent. The independent base point is located at the lower left corner of the chair. Thus, the chair rotates about the lower left corner.



In the following example, each of the three rectangles in the dynamic block reference rotates about an independent base point located at the lower left corner of each rectangle. To achieve this, you assign one rotation parameter. You then add three rotate actions. Each rotate action is associated with the rotate parameter. Each rotate action is then associated with a different object and assigned a different independent base point.



You could achieve the same result by using dependent base points, each with a different base offset, for each rotation action. However, if you need to move the rectangles independently from one another (for example, with a polar or XY parameter and a move action) in the block reference, you should use independent base points for the rotate actions, or the objects won't rotate correctly.

See also:

- Add a Rotation Parameter on page 945

To add a rotate action to a block definition

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

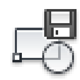
2 In the Block Editor, in the Block Authoring Palettes window, Actions tab, click the Rotate Action tool.

3 In the Block Editor drawing area, select a rotation parameter to associate with the action. (You can associate only a rotation parameter with a rotate action.)

4 Select the objects (the selection set) to associate with the action. Press Enter.

5 (Optional) Follow the Command prompts to specify the type of base point.

6 Specify the location of the action.

7 Click Block Editor tab ➤ Open/Save panel ➤ Save Block. 

8 If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard 
Command entry: BACTION

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BACTION

Adds an action to a dynamic block definition.

BACTIONSET

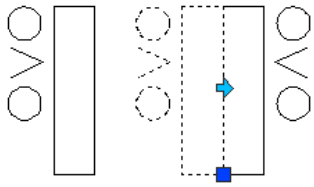
Specifies the selection set of objects associated with an action in a dynamic block definition.

BASSOCIATE

Associates an action with a parameter in a dynamic block definition.

Add a Flip Action

In a dynamic block reference, a flip action causes its associated selection set to flip about an axis called a *reflection line* when the associated parameter is edited through a grip or the Properties palette.



In a dynamic block definition, you can associate a flip action only with a flip parameter on page 946. The flip action is associated with the entire parameter, not a key point on the parameter. After associating a flip action with a flip parameter, you associate the action with a selection set of geometry. Only the selected objects will flip about the reflection line.

For example, in the following illustration, the polyline between the two circles is not included in the flip action's selection set. When the block reference is flipped, the polyline does not flip with the rest of the geometry.

See also:

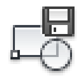
- Add a Flip Parameter on page 946

To add a flip action to a block definition

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. In the Block Editor, in the Block Authoring Palettes window, Actions tab, click the Flip Action tool.
3. In the Block Editor drawing area, select a parameter to associate with the action. (You can associate only a flip parameter with a flip action.)

4 Select the objects (the selection set) to associate with the action. Press Enter.

5 Specify the location of the action.

6 Click Block Editor tab ➤ Open/Save panel ➤ Save Block. 

7 If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard 
Command entry: BACTION

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BACTION

Adds an action to a dynamic block definition.

BACTIONSET

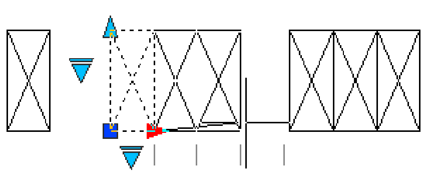
Specifies the selection set of objects associated with an action in a dynamic block definition.

BASSOCIATE

Associates an action with a parameter in a dynamic block definition.

Add an Array Action

An array action causes its associated objects to copy and array in a rectangular pattern.



In a dynamic block definition, you can associate an array action with any of the following parameters:

- Linear on page 940
- Polar on page 942
- XY on page 944

After associating an array action with a parameter, you associate the action with a selection set of geometry.

Specify Rows and Columns for an Array Action

When you associate an array action with a linear or polar parameter, you specify the column offset for the arrayed objects. The column offset determines the distance between the arrayed objects. When you edit the parameter in the block reference, the distance of the parameter (from base point to second point) is divided by the column offset to determine the number of columns (the number of objects).

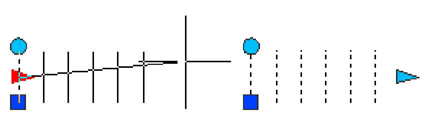
For example, you might associate an array action with a linear parameter. You specify that the array action has a column offset of 2. When you change the linear parameter in the dynamic block reference to a distance of 10, the number of columns for the block reference is 5.

If you associate an array action with an XY parameter, you can also specify the row offset.

Including Parameters in an Array Action's Selection Set

When you include a parameter in an array action's selection set, it has no effect on the behavior of the block reference. The parameter is not copied with the other objects in the selection set. Additional grips are not displayed in the block reference.

In the following example, a parking lot block can be arrayed to contain any number of spaces. The vertical line can also be rotated. Notice that even after the block has been arrayed, it still contains only one rotation grip.

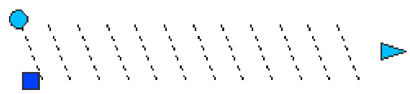


However, when you edit the grip for the parameter included in the array action's selection set, the parameter's associated action is triggered for all instances of the objects. The same behavior occurs when the parameter is not included in the array action's selection set.

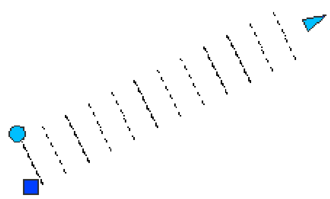
Using Rotate and Array Actions in the Same Dynamic Block

A dynamic block can contain an array action and a rotate action that have the same selection set. The order in which the block reference is arrayed and rotated affects the display of the block.

When you rotate the block first and then array it, all instances of the arrayed objects are individually rotated around their own base point.



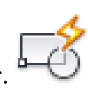
When you array the block first and then rotate it, all instances of the arrayed objects are rotated around a single base point.



See also:

- Add a Linear Parameter on page 940
- Add a Polar Parameter on page 942
- Add an XY Parameter on page 944

To add an array action to a block definition

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. In the Block Editor, in the Block Authoring Palettes window, Actions tab, click the Array Action tool.
3. In the Block Editor drawing area, select a parameter to associate with the action. (You can associate linear, polar, and XY parameters with an array action.)

4 Select the objects (the selection set) to associate with the action. Press Enter.

5 If you are associating the array action with an XY parameter, do one of the following:

- Enter the distance between rows.
- Specify a unit cell by entering two values separated by a comma or by picking two opposite points of a rectangle in the drawing area, and then go to step 7.

6 Enter a value for the distance between columns.

7 If you are assigning the array action to an XY parameter, enter the distance between rows.

8 Specify the location of the action.

9 Click Block Editor tab ➤ Open/Save panel ➤ Save Block.

10 If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard
Command entry: BACTION

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BACTION

Adds an action to a dynamic block definition.

BACTIONSET

Specifies the selection set of objects associated with an action in a dynamic block definition.

BASSOCIATE

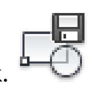
Associates an action with a parameter in a dynamic block definition.

Add a Lookup Action

A lookup action assigns custom properties and values to a dynamic block.

When you add a lookup action to a dynamic block definition and associate it with a lookup parameter, it creates a lookup table. For information about adding lookup tables to dynamic blocks, see Use Lookup Tables to Assign Data to Dynamic Blocks on page 993.

To add a lookup action to a dynamic block definition

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. In the Block Editor, in the Block Authoring Palettes window, Actions tab, click the Lookup Action tool.
3. In the Block Editor drawing area, select one or more lookup parameters to associate with the action. (You can associate only a lookup action with lookup parameters.)
4. Specify the location of the action.
5. In the Property Lookup Table dialog box, complete the table as necessary.
6. Click OK.
7. Click Block Editor tab ➤ Open/Save panel ➤ Save Block. 
8. If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard 
Command entry: BACTION

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BACTION

Adds an action to a dynamic block definition.

BACTIONSET

Specifies the selection set of objects associated with an action in a dynamic block definition.

BASSOCIATE

Associates an action with a parameter in a dynamic block definition.

BLOOKUPTABLE

Displays or creates a lookup table for a dynamic block definition.

Use Parameter Sets

You add a parameter set to your block the same way you add a parameter. The action included in the parameter set is automatically added to your block definition and associated with the added parameter. You must then associate a selection set (geometry) with each action.

When you first add a parameter set to your dynamic block definition, a yellow alert icon is displayed next to each action. This indicates that you need to associate a selection set with each action. You can double click the yellow alert icon (or use the BACTIONSET command) and follow the Command prompts to associate the action with a selection set.

NOTE When you insert a lookup parameter set and double click the yellow alert icon, the Property Lookup Table dialog box is displayed. Lookup actions are associated with the data you add to this table, not a selection set.

The following table lists the parameter sets provided on the Parameter Sets tab of the Block Authoring palette.

Parameter Set	Description
Point Move	Adds a point parameter with one grip and an associated move action to the dynamic block definition.
Linear Move	Adds a linear parameter with one grip and an associated move action to the dynamic block definition.

Parameter Set	Description
Linear Stretch	Adds a linear parameter with one grip and an associated stretch action to the dynamic block definition.
Linear Array	Adds a linear parameter with one grip and an associated array action to the dynamic block definition.
Linear Move Pair	Adds a linear parameter with two grips and a move action associated with each grip to the dynamic block definition.
Linear Stretch Pair	Adds a linear parameter with two grips and a stretch action associated with each grip to the dynamic block definition.
Polar Move	Adds a polar parameter with one grip and an associated move action to the dynamic block definition.
Polar Stretch	Adds a polar parameter with one grip and an associated stretch action to the dynamic block definition.
Polar Array	Adds a polar parameter with one grip and an associated array action to the dynamic block definition.
Polar Move Pair	Adds a polar parameter with two grips and a move action associated with each grip to the dynamic block definition.
Polar Stretch Pair	Adds a polar parameter with two grips and a stretch action associated with each grip to the dynamic block definition.
XY Move	Adds an XY parameter with one grip and an associated move action to the dynamic block definition.
XY Move Pair	Adds an XY parameter with two grips and move action associated with each grip to the dynamic block definition.
XY Move Box Set	Adds an XY parameter with four grips and a move action associated with each grip to the dynamic block definition.
XY Stretch Box Set	Adds an XY parameter with four grips and a stretch action associated with each grip to the dynamic block definition.

Parameter Set	Description
XY Array Box Set	Adds an XY parameter with four grips and an array action associated with each grip to the dynamic block definition.
Rotation	Adds a rotation parameter with one grip and an associated rotate action to the dynamic block definition.
Flip	Adds a flip parameter with one grip and an associated flip action to the dynamic block definition.
Visibility	Adds a visibility parameter with one grip. No action is required with a visibility parameter.
Lookup	Adds a lookup parameter with one grip and a lookup action to the dynamic block definition.

See also:

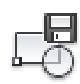
- Use Block Authoring Palettes on page 878

To add parameter set to a dynamic block definition

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 In the Block Editor, in the Block Authoring Palettes window, on the Parameter Sets tab, click a parameter set.

3 Follow the Command prompts.

4 Click Block Editor tab ➤ Open/Save panel ➤ Save Block. 

5 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

Specify Grips for Dynamic Blocks

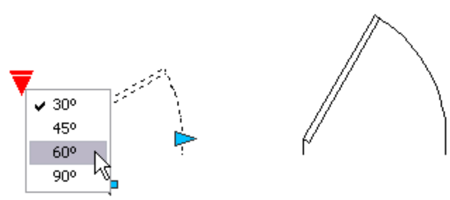
When you add a parameter to a dynamic block definition, custom grips associated with key points of the parameter are automatically added to the block.

All parameters (except the alignment parameter, which always displays one grip) have a property called *Number of Grips*. When you select a parameter in the Block Editor, the Number of Grips property is displayed in the Properties palette. This property allows you to specify, from a preset list, the number of grips you want to display for the parameter.

NOTE Parameters that are not associated with an action do not display grips.

If you specify that a parameter has 0 grips, you can still edit the dynamic block reference through the Properties palette (if the block is defined that way).

If a dynamic block definition contains visibility states or a lookup table on page 993, you can define the block so that the only grip that is displayed is a lookup grip. When you click this grip on the block reference, a drop-down list is displayed. When you select an item from the list, the display of the block reference may change.



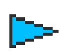
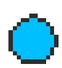
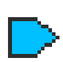
Grips are automatically added at key points on the parameter. You can reposition a grip anywhere in the block space relative to its associated key point on the parameter. When you reposition a grip, it is still tied to the key point with which it is associated. No matter where the grip is shown in the block reference, it will still manipulate the key point with which it is associated. If you move or change the key point of the parameter, the grip's position relative to the key point is maintained. Because you use grips to manipulate dynamic block references in a drawing, you should make sure that each grip is placed in a logical location. If a grip is not in a logical place, the behavior may not be what is expected.

NOTE Grips are not displayed for key points that are not associated with an action.

Both linear and polar parameters can display two, one, or no grips. When you define a linear or polar parameter to display one grip, it is displayed at the endpoint of the parameter. You should only assign actions to the endpoint of either of these parameters if you plan to display only one grip. Otherwise, you will not be able to manipulate the reference (trigger the actions) because a grip won't be displayed for the key point associated with the action.

If you reposition the grips for a dynamic block, you can use the BGRIPSET command to reset the grips to their default locations.

The type of parameter you add to the dynamic block definition determines the type of grips that are added to the block. These grips give visual clues as to how the block reference can be manipulated in a drawing. The following table details the types of grips used in dynamic blocks, what they look like, and the parameters with which they are associated.

Grip Type		How the Grip Can Be Manipulated in a Drawing	Associated Parameters
Standard		Within a plane in any direction	Base, Point, Polar, and XY
Linear		Back and forth in a defined direction or along an axis	Linear
Rotation		Around an axis	Rotation
Flip		Click to flip the dynamic block reference	Flip
Alignment		Within a plane in any direction; when moved over an object, triggers the block reference to align with the object	Alignment
Lookup		Click to display a list of items	Visibility, Lookup

Specify Tooltips on Grips

Each of the dynamic block parameters has one or more description fields with the exception of the basepoint and alignment parameters. These descriptions function as tooltips or prompts for the associated parameter, and their display is controlled by theGRIPTIPS system variable.

The following is a table of dynamic block parameters and available descriptions.

Parameter	Description
Point	Position description
Linear	Distance description
Polar	Distance description, angle description
XY	Horizontal distance description, vertical distance description
Rotation	Angle description
Flip	Flip description
Visibility	Visibility description
Lookup	Lookup description
Alignment	Set to “Aligns block to object”
Basepoint	No special tooltip is required

Specify Insertion Cycling for Grips in Dynamic Blocks

Grips in dynamic blocks have a property called *Cycling*. When this property is set to Yes, the grip becomes an available insertion point for the dynamic block reference. You can use the BCYCLEORDER command to turn on and off cycling for grips in dynamic blocks as well as specify the cycling order for the grips. When you insert the dynamic block reference in a drawing, you can use the Ctrl key to cycle through the available grips to select which grip will be the insertion point for the block.

To specify the number of custom grips displayed for a parameter in a dynamic block

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 In the Block Editor, select a parameter.

3 On the Properties palette, under Misc, click Number of Grips, and then select the number of grips you want to display for the parameter.

4 Click Block Editor tab ➤ Open/Save panel ➤ Save Block.

5 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Command entry: BGRIPSET
Shortcut menu: In the Block Editor, select a parameter. Right-click in the drawing area. Click Grip Display ➤ *any value*.

To reposition a grip in a dynamic block definition

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 In the Block Editor, select a custom grip (not the standard grip associated with the key point of the parameter).

3 Do one of the following to reposition the grip:
   - Drag the grip to another position in the block space.
   - On the Properties palette, under Geometry, enter values for the Base X and Base Y properties. (These values are relative to the base point of the parameter.)

4 Click Block Editor tab ➤ Open/Save panel ➤ Save Block.

5 (Optional) If you are finished using the Block Editor, click Close Block Editor.

To reset grips in a dynamic block definition to their default locations

1. Click Insert tab ➤ Block panel ➤ Block Editor.
2. In the Block Editor, at the Command prompt, enter **bgripset**.
3. Select the grip for which you want to reset the grip position(s).
4. At the Command prompt, enter **reposition**.
5. Click Block Editor tab ➤ Open/Save panel ➤ Save Block.
6. (Optional) If you are finished using the Block Editor, click Close Block Editor.

Shortcut menu: In the Block Editor, select a parameter. Right-click in the drawing area. Click Grip Display ➤ *Reset Position*.

To turn insertion cycling on or off for a custom grip in a dynamic block reference

1. Click Insert tab ➤ Block panel ➤ Block Editor.
2. In the Block Editor, at the Command prompt, enter **bcycleorder**.
3. In the Insertion Cycling Order dialog box, select a grip from the list, and click Cycling to turn cycling on or off for the grip. (A check mark in the Cycling column indicates that cycling is turned on for the grip.)
4. Click OK.
5. Click Block Editor tab ➤ Open/Save panel ➤ Save Block.
6. (Optional) If you are finished using the Block Editor, click Close Block Editor.

Shortcut menu: In the Block Editor, select a dynamic grip. Right-click in the drawing area. Click Insertion Cycling.

To modify the insertion cycling order for custom grips in a dynamic block reference

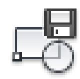
1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 In the Block Editor, enter **bcycleorder** at the Command prompt.

3 In the Insertion Cycling Order dialog box, select a grip from the list, and click Move Up or Move Down. (A check mark in the Cycling column indicates that cycling is turned on for the grip.)

4 Repeat step 2 until you are finished modifying the grip cycling order.

5 Click OK.

6 Click Block Editor tab ➤ Open/Save panel ➤ Save Block. 

7 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BCYCLEORDER

Changes the cycling order of grips for a dynamic block reference.

BGRIPSET

Creates, deletes, or resets grips associated with a parameter.

BPARAMETER

Adds a parameter with grips to a dynamic block definition.

System Variables

GRIPTIPS

Controls the display of grip tips when the cursor hovers over grips on dynamic blocks and custom objects that support grip tips.

Use Lookup Tables to Assign Data to Dynamic Blocks

You can use a lookup table to define properties for and assign property values to a dynamic block.

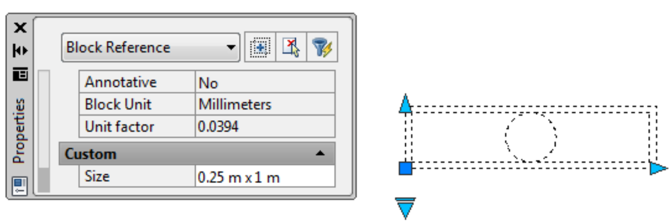
Overview of Using Lookup Tables to Assign Data to Dynamic Blocks

You can use a *lookup table* to define properties for and assign property values to a dynamic block. Using lookup tables is a powerful way to associate parameter values for the dynamic block reference with other data that you specify (for example, a model or part number). You can extract this data from block references in a drawing just as you would extract block attribute data.

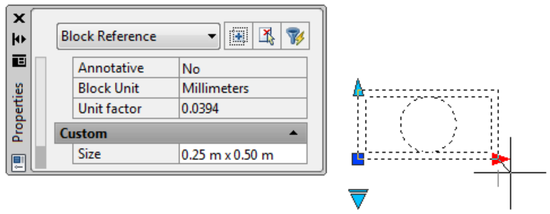
NOTE Constraint parameters cannot be added to a lookup table. Instead, you should use a Block Properties Table. For more information, see Use a Block Properties Table on page 876.

When completed, the lookup table assigns property values to the dynamic block reference based on how it is manipulated in a drawing. Conversely, you can change how the block reference is displayed in the drawing by changing the value of a lookup property of the block reference through a lookup grip or the Properties palette.

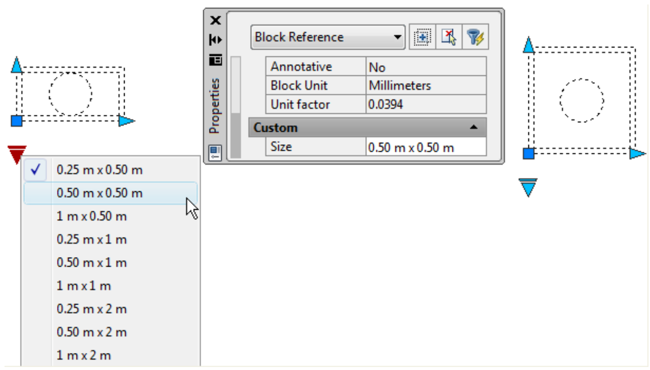
The following example shows a dynamic block reference for a lighting fixture. For this block, the Custom area of the Properties palette displays a size property. This property is defined in the lookup table for the block.



If you change the size of the lighting fixture in your drawing (using the custom grips), the size property in the Properties palette changes accordingly.



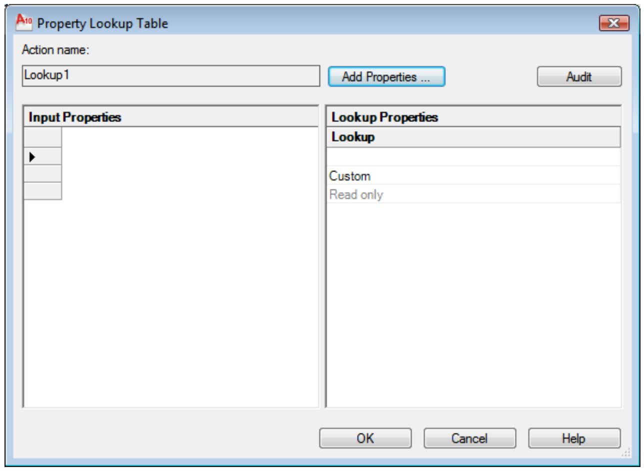
You can also use the lookup grip to change the display of the block reference. When you select a size from the list that is displayed when you click the grip, the block's geometry changes and the new size is displayed in the Properties palette under Custom.



Create a Lookup Table

After you've drawn the geometry and added the appropriate parameters and actions required for the dynamic block's functionality, you can add a lookup table to the block definition.

A blank lookup table is created when you add a lookup parameter to a dynamic block definition and then add a lookup action and associate it with the lookup parameter. The lookup table is displayed in the lookup table dialog box.



Add Properties and Values to a Lookup Table

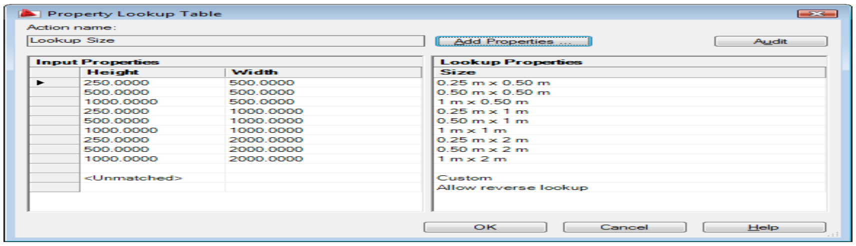
The Property Lookup Table consists of input properties and lookup properties. Each input property and lookup property is represented by a column in the table.

- **Input properties.** Parameters other than lookup parameters (for example, a linear parameter labeled "Width"). You can create a column under Input Properties for every parameter in the block definition except for lookup, alignment, and base point parameters.
- **Lookup properties.** Lookup parameters. You add one lookup parameter to the dynamic block definition for every lookup property column you want to add to the lookup table. The lookup parameter label is used as the property name.

After you add the properties (columns) to the table, you add values to the cells in each column. You click a cell and enter a value. Make sure to follow the guidelines in Specify Values for Lookup Tables on page 1001.

If you defined a value set for an input property (parameter) in the table, a drop-down list of available values will display when you click a cell in that column.

The following lookup table is used for the lighting fixture block shown previously. The linear parameters that have been labeled "Height" and "Width" have been added as input properties. "Size" has been added as a lookup property. The values under Height and Width are taken from the value sets defined for each of those parameters. The values under Size are entered manually.



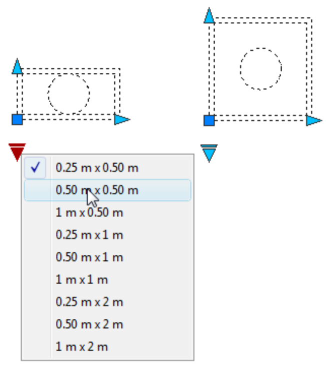
When the parameter values in a dynamic block reference match a row of input property values, the corresponding lookup property values in that table row are assigned to the block reference. These lookup properties and values are displayed in the Properties palette under Custom.

For example, when you edit the lighting fixture block reference to have a height of 250 centimeters and a width of 500 centimeters, the Size property in the Custom area of the Properties palette will display a value of 0.25 m x 0.50 m.

In the lookup table, at the bottom of the input properties is a row labeled *<Unmatched>*. When the parameter values in a block reference do not match a row of input property values defined in the block's lookup table, the lookup property value that corresponds to <Unmatched> is assigned to the block reference. The default value is *Custom*, but you can specify another value by changing it in the table.

Enable Reverse Lookup

You can also specify that a lookup property allows for reverse lookup. This adds a lookup grip to the dynamic block reference. When you click this grip, a drop-down list of the lookup values for that lookup property (column in the table) is displayed. When you select a value from the list, the corresponding input property values are assigned to the block reference. Depending on how the block was defined, this usually results in a change in the block reference's geometry.



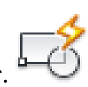
To enable reverse lookup for a lookup property, each row in the lookup table must be unique. After you add properties and values to the table, you can check for errors and empty cells by clicking the audit button in the Property Lookup Table dialog box. You can also use the shortcut menu options to insert, delete, or reorder rows.

NOTE If you leave a cell empty in the lookup properties column and the cell has an associated input property in the same row, the lookup property will not allow for reverse lookup.

See also:

- Extract Data from Block Attributes on page 1034
- Extract Block Attribute Data (Advanced) on page 1035

To create a lookup table for a dynamic block

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. In the Block Editor, add one or more lookup parameters to a dynamic block definition by following the steps in To add a lookup parameter to a dynamic block definition on page 958.
3. In the Block Editor, add a lookup action to a dynamic block definition by following the steps in To add a lookup action to a dynamic block definition on page 983.

   The Property Lookup Table dialog box is displayed.

4. Use any of the following procedures to set up a lookup table for the dynamic block:
   - To add input properties to a lookup table on page 998
   - To add lookup properties to a lookup table on page 998
   - To enable reverse lookup on page 999
5. Click Block Editor tab ➤ Open/Save panel ➤ Save Block.
6. (Optional) If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard
Command entry: BPARAMETER

To add input properties to a lookup table

1. In the Block Editor, double-click a lookup action in a dynamic block definition.
2. In the Property Lookup Table dialog box, click Add Properties.
3. In the Add Parameter Properties dialog box, in the bottom-left corner of the dialog box, click Add Input Properties.
4. In the Parameter Properties list, select the parameter properties you want to add to the lookup table. Hold Ctrl down to select more than one property.
5. Click OK.

Command entry: BLOOKUPTABLE

To add lookup properties to a lookup table

1. Click Insert tab ➤ Block panel ➤ Block Editor.
2. In the Block Editor, double-click a lookup action in a dynamic block definition.

3 In the Property Lookup Table dialog box, click Add Properties.

4 In the Add Parameter Properties dialog box, in the bottom-left corner of the dialog box, click Add Lookup Properties.

5 In the Parameter Properties list, select the lookup parameter properties you want to add to the lookup table. Hold Ctrl down to select more than one property.

6 Click OK.

Command entry: BLOOKUPTABLE

To add values to a lookup table

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 In the Block Editor, double-click a lookup action in a dynamic block definition.

3 In the Property Lookup Table dialog box, click an empty cell and do one of the following:

- Select a value from the drop-down list. (A drop-down list only displays if a value set has been defined for the parameter.)
- Enter a value.

4 Repeat Step 2 as required to complete the table.

5 Click OK.

Command entry: BLOOKUPTABLE

To enable reverse lookup

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 In the Block Editor, double-click a lookup action in a dynamic block definition.

3 In the Property Lookup Table dialog box, at the bottom of a Lookup Property column, click Read Only, and then select Allow Reverse Lookup from the list.

NOTE You can only select Allow Reverse Lookup if you have entered data in the column.

4 Click OK.

Command entry: BLOOKUPTABLE

To make the lookup property read-only

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 In the Block Editor, double-click a lookup action in a dynamic block definition.

3 In the Property Lookup Table dialog box, at the bottom of a Lookup Property column, click Allow Reverse Lookup, and then select Read Only from the list.

4 Click OK.

Command entry: BLOOKUPTABLE

To check for errors in a lookup table

- In the Property Lookup Table dialog box, click Audit.

To open an existing lookup table

1 Click Insert tab ➤ Block panel ➤ Block Editor.

2 In the Block Editor, double-click a lookup action.

Command entry: BLOOKUPTABLE

Quick Reference

Commands

BACTION

Adds an action to a dynamic block definition.

BEDIT

Opens the block definition in the Block Editor.

BLOOKUPTABLE

Displays or creates a lookup table for a dynamic block definition.

BPARAMETER

Adds a parameter with grips to a dynamic block definition.

Specify Values for Lookup Tables

The following rules apply when you specify values in lookup tables:

- Use a comma as the delimiter between values.
- You can specify any number of unique values separated by commas. For example: 5,6,7 5.5,6.25
- To specify a range, use brackets [ ] to specify that the range includes the values separated by a comma, or use parentheses ( ) to specify that the range does not include the values separated by a comma.
- For a continuous range, use a pair of values separated by a comma, enclosed in brackets or parentheses. For example: [3,10] specifies any value between 3 and 10, including 3 and 10 (3,10) specifies any value between 3 and 10, not including 3 and 10.
- For an open-ended range, use one value with a comma, enclosed in brackets or parentheses. For example: [,5] specifies less than or equal to 5; (5,) specifies greater than 5.
- Use no more than 256 characters in a table cell.
- You can use architectural and mechanical unit syntax (for example, 15'1/4").
- If you enter a value in an invalid format, the value will be reset to the last value when you move to another cell in the table.

Lookup tables support the following:

- All numeric parameter properties (for example, distance and angles for point, linear, polar, XY, and rotation parameters)

- Text string parameter properties (for example, flip and visibility parameter values)

Quick Reference

Commands

BLOOKUPTABLE

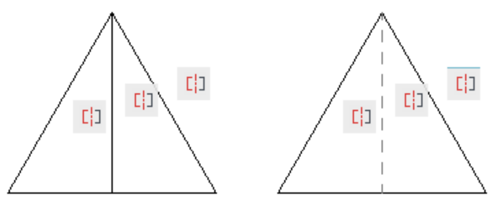
Displays or creates a lookup table for a dynamic block definition.

Create Construction Geometry Within a Block

You can convert objects to construction geometry in the Block Editor.

You can create construction geometry that will display within the Block Editor, but not in the drawing editor.

In the example below, a symmetric constraint on page 913 has been added to the block definition on the left. However, you may not want the line of symmetry to be displayed when the block is inserted into your drawing. In the example on the right, the line of symmetry has been converted to a dashed line that will not display when the block is inserted into a drawing.



You can add construction geometry (BCONSTRUCTION command) to the selection sets of legacy actions. The construction geometry is not affected by the visibility states. It is filtered from the selection set in the BVSTATE command when you add or remove geometry from a visibility state.

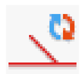
NOTE When you explode a block containing construction geometry in previous versions of AutoCAD, the geometry is hidden in the drawing.

See also:

- Draw Construction and Reference Geometry on page 840
- Add a Symmetric Constraint on page 913

To convert an object into construction geometry

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 Click Block Editor tab ➤ Geometric panel ➤ Construction Geometry.


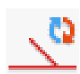
3 Select the **Convert** option.

4 Select the geometry to convert into construction geometry. Press Enter.

Command entry: BCONSTRUCTION

To revert construction geometry back to a regular object

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

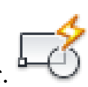
2 Click Block Editor tab ➤ Geometric panel ➤ Construction Geometry.


3 Select the **Revert** option.

4 Select the construction geometry to revert into regular geometry. Press Enter.

Command entry: BCONSTRUCTION

To display all construction geometry in the Block Editor

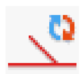
1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 Click Block Editor tab ➤ Geometric panel ➤ Construction Geometry.


3 Select the Show All option.

Command entry: BCONSTRUCTION

To hide all construction geometry in the Block Editor

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. Click Block Editor tab ➤ Geometric panel ➤ Construction Geometry.
   
3. Select the Hide All option.

Command entry: BCONSTRUCTION

Quick Reference

Commands

BCONSTRUCTION

Converts geometry into construction geometry.

BVSTATE

Creates, sets, or deletes a visibility state in a dynamic block.

System Variables

BLOCKTESTWINDOW

Indicates whether or not a test block window is current.

Specify Custom Properties for Dynamic Blocks

When you create a dynamic block, you can specify custom properties for the block. You can also specify whether or not these properties appear in the Properties palette when you select the block reference in a drawing.

Overview of Specifying Custom Properties for Dynamic Blocks

In the Block Editor, you can specify properties for a parameter in a dynamic block definition. Some of these properties can be displayed as *custom properties* for the dynamic block reference when it is in a drawing. These properties are displayed under Custom in the Properties palette.

In the Block Editor, you can specify parameter labels. When you select the dynamic block reference in a drawing, these properties are shown under Custom in the Properties palette. It is good practice to specify unique parameter labels within the block.

Depending on the parameters used in the dynamic block definition, other parameter properties may be listed under Custom in the Properties palette when you select the dynamic block reference in a drawing. For example, a polar parameter has an angle property that displays in the Properties palette. Depending on how the dynamic block is defined, these properties might display values such as size, angle, and position for a selected block reference.

You can specify whether or not these custom properties are displayed for the block reference when it is selected in a drawing. These properties can also be extracted using the Attribute Extraction wizard.

You can also use the Properties palette to specify the geometric properties (such as color, linetype, and lineweight) of a parameter. These properties are listed in the Properties Palette under Geometry when you select a parameter in the Block Editor.

Other parameter properties, such as Value Set properties and Chain Actions, define how the block reference will function in a drawing.

In the Block Editor, you can also specify if the block can be exploded and if the block can be non-uniformly scaled.

See also:

- Define User Parameters in Dynamic Blocks on page 875
- Specify Value Sets for Dynamic Blocks on page 1009
- Allow Chained Actions for Dynamic Blocks on page 1014
- Specify Grips for Dynamic Blocks on page 987
- Extract Data from Block Attributes on page 1034

To specify a parameter label

1 Ensure that the Properties palette is displayed. If not, click Tools menu ➤ Palettes ➤ Properties.

2 Click Insert tab ➤ Block panel ➤ Block Editor.

3 In the Block Editor, select a parameter.

4 In the Properties palette, under Property Labels, click *<parameter type>* label.

5 Enter a label for the parameter.

6 Click Block Editor tab ➤ Open/Save panel ➤ Save Block.

7 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard
Command entry: PROPERTIES
Shortcut menu: Right-click the selected object. Click Rename Label. Enter a new name for the parameter, and press Enter.

To specify the display of custom properties in a dynamic block reference

1 Ensure that the Properties palette is displayed. If not, click Tools menu ➤ Palettes ➤ Properties.

2 Click Insert tab ➤ Block panel ➤ Block Editor.

3 In the Block Editor, select a parameter.

4 In the Properties palette, under Misc., click Show Properties.

5 On the drop-down list, do one of the following:
   - Select Yes to display custom properties for the block reference.
   - Select No to specify that the custom properties will not display for a block reference.

6 Click Block Editor tab ➤ Open/Save panel ➤ Save Block.

7 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard
Command entry: PROPERTIES

To specify that dynamic block reference can be exploded

1 Use one of the following procedures to open a block definition in the Block Editor:
   - To open an existing block definition in the Block Editor on page 869
   - To open a drawing file saved as a block (not dynamic) in the Block Editor on page 871
   - To open a drawing file saved as a dynamic block in the Block Editor on page 871

2 Click Insert tab ➤ Block panel ➤ Block Editor.

3 In the Block Editor, make sure nothing is selected.

4 In the Properties palette, under Block, click Allow Exploding.

5 On the drop-down list, select Yes or No.

6 Click Block Editor tab ➤ Open/Save panel ➤ Save Block.

7 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard
Command entry: BEDIT

To prevent non-uniform scaling in dynamic block reference

1. Use one of the following procedures to open a block definition in the Block Editor:
   - To open an existing block definition in the Block Editor on page 869
   - To open a drawing file saved as a block (not dynamic) in the Block Editor on page 871
   - To open a drawing file saved as a dynamic block in the Block Editor on page 871
2. Click Insert tab ➤ Block panel ➤ Block Editor.
3. In the Block Editor, make sure nothing is selected.
4. In the Properties palette, under Block, click Scale Uniformly.
5. On the drop-down list, select Yes.
6. Click Block Editor tab ➤ Open/Save panel ➤ Save Block.
7. (Optional) If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard
Command entry: BEDIT

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BPARAMETER

Adds a parameter with grips to a dynamic block definition.

PROPERTIES

Controls properties of existing objects.

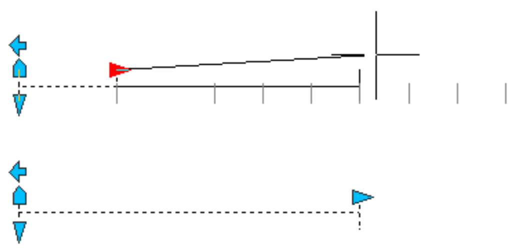
Specify Value Sets for Dynamic Blocks

A value set is a range or list of values specified for a parameter. These values can be displayed for the block reference as a drop-down list next to the parameter label under Custom in the Properties palette. When you define a value set for a parameter, the parameter is limited to these values when the block reference is manipulated in a drawing. For example, if you define a linear parameter in a block that represents a window to have a value set of 20, 40, and 60, the window can only be stretched to 20, 40, or 60 units.

When you create a value list for a parameter, the value of the parameter as it exists in the definition is automatically added to the value set. This is the default value for the block reference when you insert it in a drawing.

In a block reference, if you change the parameter's value to a value other than one in the list, the parameter will adjust to the closest valid value. For example, you define a linear parameter to have a value set of 2, 4, and 6. When you try to change the value of that parameter in a block reference to 10, the resulting parameter value is 6 because it is the closest valid value.

When you specify a value set for a parameter in a dynamic block, tick marks are displayed when you grip-edit the block reference in a drawing. The tick marks indicate the locations for the parameter's valid values.



NOTE If you redefine the values in a value set after you've added the parameter properties to a lookup table, make sure to update the lookup table to match the new values in the value set.

To specify a value set for a linear or polar parameter

1 Ensure that the Properties palette is displayed. If not, click Tools menu ➤ Palettes ➤ Properties.

2 Click Insert tab ➤ Block panel ➤ Block Editor. 

3 In the Block Editor, select a linear or polar parameter.

4 In the Properties palette, under Value Set, click Dist Type.

5 On the drop-down list, select List.

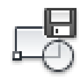
6 In the Properties palette, under Value Set, click Dist Value List.

7 Click the [...] button.

8 In the Add Distance Value dialog box, under Distances to Add, enter one value or two or more values separated by commas.

9 Click Add.

10 Click OK.

11 Click Block Editor tab ➤ Open/Save panel ➤ Save Block. 

12 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard
Command entry: PROPERTIES

To specify a value set for an XY parameter

1 Ensure that the Properties palette is displayed. If not, click Tools menu ➤ Palettes ➤ Properties.

2 Click Insert tab ➤ Block panel ➤ Block Editor. 

3 In the Block Editor, select an XY parameter.

4 In the Properties palette, under Value Set, click Hor Type or Ver Type.

5 On the drop-down list, select List.

6 In the Properties palette, under Value Set, click Hor Value List or Ver Value List.

7 Click the [...] button.

8 In the Add Distance Value dialog box, under Distances to Add, enter one value or two or more values separated by commas.

9 Click Add.

10 Click OK.

11 Click Block Editor tab ➤ Open/Save panel ➤ Save Block.

12 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard
Command entry: PROPERTIES

To specify a value set for a rotation parameter

1 Ensure that the Properties palette is displayed. If not, click Tools menu ➤ Palettes ➤ Properties.

2 Click Insert tab ➤ Block panel ➤ Block Editor.

3 In the Block Editor, select a rotation parameter.

4 In the Properties palette, under Value Set, click Ang Type.

5 On the drop-down list, select List.

6 In the Properties palette, under Value Set, click Ang Value List.

7 Click the [...] button.

8 In the Add Angle Value dialog box, under Angle to Add, enter one value or two or more values separated by commas.

9 Click Add.

10 Click OK.

11 Click Block Editor tab ➤ Open/Save panel ➤ Save Block.

12 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard 
Command entry: PROPERTIES

To delete values from a parameter's value set

1 Ensure that the Properties palette is displayed. If not, click Tools menu ➤ Palettes ➤ Properties.

2 Click Insert tab ➤ Block panel ➤ Block Editor. 

3 In the Block Editor, select a linear, polar, XY, or rotation parameter.

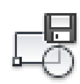
4 In the Properties palette, under Value Set, click one of the following:
- Dist Value List
- Ang Value List
- Hor Value List
- Ver Value List

5 Click the [...] button.

6 In the Add Distance Value or Add Angle Value dialog box, select a value from the list to delete.

7 Click Delete.

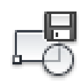
8 Click OK.

9 Click Block Editor tab ➤ Open/Save panel ➤ Save Block. 

10 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard 
Command entry: PROPERTIES

To specify an incremental value set for a linear, polar, XY, or rotation parameter

1. Ensure that the Properties palette is displayed. If not, click Tools menu ➤ Palettes ➤ Properties.
2. Click Insert tab ➤ Block panel ➤ Block Editor. 
3. In the Block Editor, select a linear, polar, XY, or rotation parameter.
4. In the Properties palette, under Value Set, click Dist Type, Ang Type, Hor Type, or Ver Type.
5. On the drop-down list, select Increment.
6. In the Properties palette, under Value Set, click Dist Increment, Ang Increment, Hor Increment, or Ver Increment, and then enter an incremental value for the parameter.
7. Click Dist Minimum, Ang Minimum, Hor Minimum, or Ver Minimum, and then enter a minimum value for the parameter.
8. Click Dist Maximum, Ang Maximum, Hor Maximum, or Ver Maximum, and then enter a maximum value for the parameter.
9. Click Block Editor tab ➤ Open/Save panel ➤ Save Block. 
10. (Optional) If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard 
Command entry: PROPERTIES

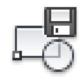
To specify minimum and maximum values for a linear, polar, XY, or rotation parameter

1. Ensure that the Properties palette is displayed. If not, click Tools menu ➤ Palettes ➤ Properties.
2. Click Insert tab ➤ Block panel ➤ Block Editor. 

3 In the Block Editor, select a linear, polar, XY, or rotation parameter.

4 In the Properties palette, under Value Set, click Dist Minimum, Ang Minimum, Hor Minimum, or Ver Minimum, and then enter a minimum value for the parameter.

5 Click Dist Maximum, Ang Maximum, Hor Maximum, or Ver Maximum, and then enter a maximum value for the parameter.

6 Click Block Editor tab ➤ Open/Save panel ➤ Save Block. 

7 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard 
Command entry: PROPERTIES

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BPARAMETER

Adds a parameter with grips to a dynamic block definition.

PROPERTIES

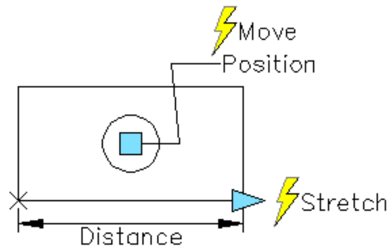
Controls properties of existing objects.

Allow Chained Actions for Dynamic Blocks

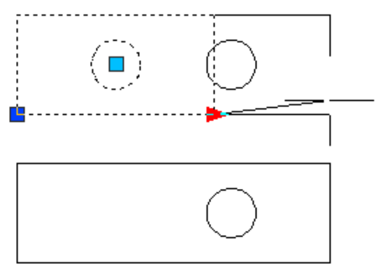
Point, linear, polar, XY, and rotation parameters have a property called *Chain Actions*. This property affects the parameter behavior if the parameter is part of an action's selection set.

For example, you might include a point parameter in the selection set of a stretch action that is associated with a linear parameter. When the linear parameter is edited in a block reference, its associated stretch action triggers a change in its selection set. Because the point parameter is included in the selection set, the point parameter is edited by the change in the linear parameter.

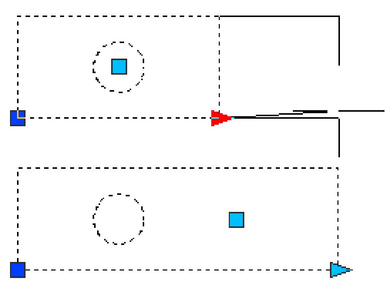
The following example shows a block definition in the Block Editor. The point parameter (labeled Position) is included in the stretch action's selection set.



If the Chain Actions property for the point parameter is set to Yes, a change in the linear parameter will trigger the move action associated with the point parameter, just as if you edited the point 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 move action is not triggered by the changes to the linear parameter. Thus, the circle doesn't move.



To specify the Chain Actions property for a point, linear, polar, XY, or rotation parameter

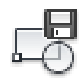
1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 In the Block Editor, click Tools menu ➤ Properties. The Properties palette is displayed.

3 In the Block Editor, select a point, linear, polar, XY, or rotation parameter.

4 In the Properties palette, under Misc, click Chain Actions.

5 On the drop-down list, select Yes or No.

6 Click Block Editor tab ➤ Open/Save panel ➤ Save Block. 

7 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Toolbar: Standard 
Command entry: PROPERTIES

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

BPARAMETER

Adds a parameter with grips to a dynamic block definition.

PROPERTIES

Controls properties of existing objects.

Use Distance Multiplier and Angle Offset Action Overrides

Action overrides are properties of actions that have no effect on the block reference until it is manipulated in a drawing. Use distance multiplier overrides with the following actions:

- Move on page 962
- Stretch on page 968
- Polar Stretch on page 972

You can specify these action override properties by following the Command prompts when you add an action to a dynamic block definition. You can also specify these properties in the Properties palette when you select an action in the Block Editor.

Distance Multiplier Overrides

Use the distance multiplier property to change a parameter value by a specified factor. For example, if you set the distance multiplier property to 2 for a stretch action, the associated geometry in the block reference would increase and double the distance of the grip movement.

Angle Offset Overrides

Use the angle offset property to increase or decrease the angle of a changed parameter value by a specified amount. For example, if you set the angle offset property of a move action to 90, the block reference would move 90 degrees beyond the angle value of the grip movement.

See also:

- Add a Move Action on page 962
- Add a Stretch Action on page 968
- Add a Polar Stretch Action on page 972

To specify a distance multiplier override for an action

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 In the Block Editor, select a move, stretch, polar stretch, or array action.

3 On the Properties palette, Overrides area, Distance Multiplier, enter a value. Press Enter.

To specify an angle offset override for an action

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 In the Block Editor, select a move, stretch, or polar stretch action.

3 On the Properties palette, Overrides area, Angle Offset, enter a value. Press Enter.

Quick Reference

Commands

BACTION

Adds an action to a dynamic block definition.

BEDIT

Opens the block definition in the Block Editor.

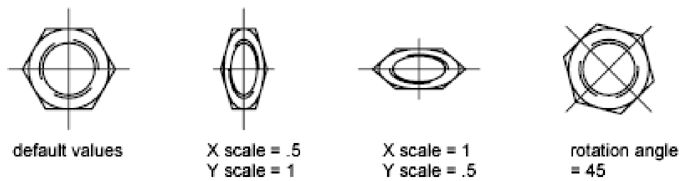
PROPERTIES

Controls properties of existing objects.

Insert Blocks

When you insert a block, you create a block reference and specify its location, scale, and rotation.

You can specify the scale of a block reference using different *X, Y,* and *Z* values. Inserting a block creates an object called a block reference because it references a block definition stored in your current drawing. An orange lightning bolt icon displayed in the bottom-right corner of the block preview indicates that the block is dynamic.



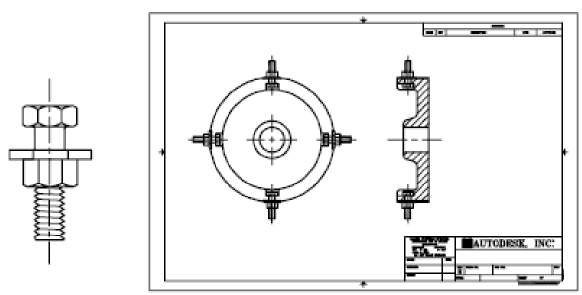
If you insert a block that uses different drawing units than the units specified for the drawing, the block is automatically scaled by a factor equivalent to the ratio between the two units.

If you insert a block reference that includes editable custom properties or attributes, you can change the values of these custom properties and attributes in the Properties palette while you insert the block. The block's custom properties and attributes become editable in the Properties palette after the block is specified in one of the following ways:

- In the Insert dialog box after clicking OK.
- Using the -INSERT command after entering the block name.
- Clicking a block tool on a tool palette.

Insert a Drawing File as a Block

When you insert an entire drawing file into another drawing, the drawing information is copied into the block table of the current drawing as a block definition. Subsequent insertions reference the block definition with different position, scale, and rotation settings, as shown in the following illustration.



fastener

block instances of fastener

Xrefs contained in a drawing you insert may not be displayed properly unless the xref was previously inserted or attached to the destination drawing.

Insert Blocks from Tool Palettes

You can insert blocks from tool palettes by dragging the block tool into the drawing or by clicking the block tool and then specifying an insertion point.

You can choose to be prompted for a rotation angle (starting from 0) when you click and place the block. When you select this option, the angle that is specified under Rotation in the Tool Properties dialog box is ignored. The prompt for a rotation angle is not shown if you drag the block or xref or, if at the initial insertion Command prompt, you enter **rotate**.

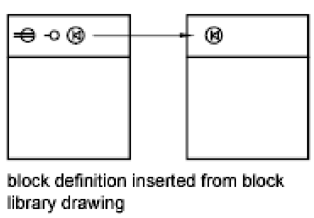
Blocks that are placed by dragging from a tool palette must often be rotated or scaled after they are placed. You can use object snaps when dragging blocks from a tool palette; however, grid snap is suppressed during dragging.

When a block is dragged from a tool palette into a drawing, it is scaled automatically according to the ratio of units defined in the block and defined in the current drawing. For example, if the current drawing uses meters as its units and a block is defined using centimeters as its units, the ratio of the units is 1 m/100 cm. When the block is dragged into the drawing, it is inserted at 1/100 scale.

NOTE In the Options dialog box, User Preferences tab, the Source Content Units and Target Drawing Units settings are used when Drag-and-Drop Scale is set to Unitless, either in the source block or target drawing.

Insert Blocks from Block Libraries

You can insert one or more block definitions from an existing drawing file into your current drawing file. Choose this method when retrieving blocks from block library drawings. A block library drawing contains block definitions of symbols with similar functions. These block definitions are stored together in a single drawing file for easy accessibility and management.



Insert Blocks with DesignCenter

Use DesignCenter to insert blocks from the current drawing or from another drawing. Drag and drop the block names for quick placement. Double-click the block names to specify the precise location, rotation, and scale of the blocks.

You cannot add blocks to a drawing while another command is active, and you can only insert or attach one block at a time.

Insert Blocks at Intervals

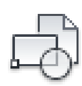
You can insert blocks at intervals along a selected geometric object.

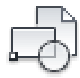
- Use MEASURE to insert a block at measured intervals.
- Use DIVIDE to insert a block at proportional (evenly spaced) intervals.

See also:

- Create Drawing Files for Use as Blocks on page 858
- Overview of Blocks on page 853
- Create Block Libraries on page 864
- Work with Dynamic Blocks in Drawings on page 1023
- Add Text and Blocks to Tables on page 1606
- Add Content with DesignCenter on page 96
- Create and Use Tools from Objects and Images on page 56

To insert a block defined in the current drawing

1. Click Home tab ➤ Block panel ➤ Insert. 
2. In the Insert dialog box, in the Name box, select a name from a list of block definitions.
3. If you want to use the pointing device to specify the insertion point, scale, and rotation, select Specify On-Screen. Otherwise, enter values in the Insertion Point, Scale, and Rotation boxes.
4. If you want the objects in the block to be inserted as individual objects instead of as a single block, select Explode.
5. Click OK.

Toolbar: Insert 
Command entry: INSERT

To insert a drawing file as a block by dragging

1. From Windows Explorer or any folder, drag the drawing file icon into the drawing area.

   When you release the button, you are prompted for an insertion point.

2. Specify the insertion point and scale and rotation values.

To insert a block using DesignCenter

1. If DesignCenter is not already open, click Tools menu ➤ Palettes ➤ DesignCenter.
2. Do one of the following to list the content you want to insert:
   - On the DesignCenter toolbar, click Tree View Toggle. Click the folder that contains the drawing you want to insert.
   - Click the icon of a drawing file displayed in the tree view.
3. Do one of the following to insert the content:
   - Drag the drawing file or block into your current drawing. Use this option when you want to insert blocks quickly and move or rotate the blocks to their precise locations later.
   - Double-click the drawing file or block that you want to insert into your current drawing. Use this option when you want to specify the exact placement, rotation, and scale of the block as you insert it. Use this option also when you want to update a block reference in your drawing from the original source drawing file.

Toolbar: Standard 
Command entry: ADCENTER

Quick Reference

Commands

ADCENTER

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

DIVIDE

Creates evenly spaced point objects or blocks along the length or perimeter of an object.

INSERT

Inserts a block or drawing into the current drawing.

MEASURE

Creates point objects or blocks at measured intervals along the length or perimeter of an object.

System Variables

ATTDIA

Controls whether the INSERT command uses a dialog box for attribute value entry.

INSNAME

Sets a default block name for the INSERT command.

INSUNITS

Specifies a drawing-units value for automatic scaling of blocks, images, or xrefs when inserted or attached to a drawing.

INSUNITSDEFSOURCE

Sets source content units value when INSUNITS is set to 0.

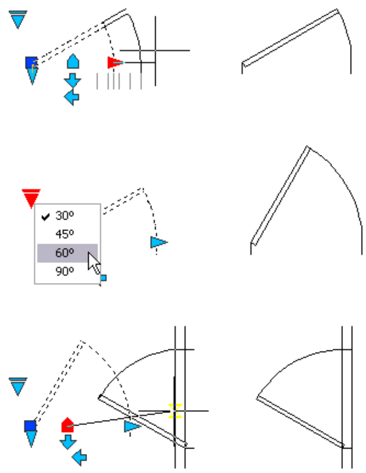
INSUNITSDEFTARGET

Sets target drawing units value when INSUNITS is set to 0.

Work with Dynamic Blocks in Drawings

A *dynamic* block has flexibility and intelligence. A dynamic block reference can easily be changed in a drawing while you work. You can manipulate the geometry through custom grips or custom properties. This allows you to adjust the block reference in-place as necessary rather than searching for another block to insert or redefining the existing one.

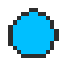
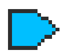
For example, if you insert a door block reference in a drawing, you might need to change the size of the door while you're editing the drawing. If the block is dynamic and defined to have an adjustable size, you can change the size of the door simply by dragging the custom grip or by specifying a different size in Properties palette. You might also need to change the open angle of the door. The door block might also contain an alignment grip, which allows you to align the door block reference easily to other geometry in the drawing.



Hovering over a grip can also display a tooltip or prompt that explains the parameter related to the grip. The display of the tooltip is controlled by theGRIPTIPS system variable.

A dynamic block can have custom grips and custom properties. Depending on how the block was defined, you may be able to manipulate the block through these custom grips and custom properties. By default, custom grips for a dynamic block are a different color than standard grips. You can change the display color for custom grips with theGRIPDYNCOLOR system variable. The following table shows the different types of custom grips that can be included in a dynamic block.

Grip Type		How the Grip Can Be Manipulated in a Drawing
Standard		Within a plane in any direction

Grip Type		How the Grip Can Be Manipulated in a Drawing
Linear		Back and forth in a defined direction or along an axis
Rotation		Around an axis
Flip		Clicked to flip the dynamic block reference
Alignment		Within a plane in any direction; when moved over an object, triggers the block reference to align with the object
Lookup		Clicked to display a list of items

After you've manipulated a dynamic block in a drawing, you can reset it. When you reset a block reference, the block changes back to the default specified in the block definition. If you non-uniformly scale or explode a dynamic block reference, it loses its dynamic properties. You can reset the block to its default values, which will make it dynamic again.

Some dynamic blocks are defined so that geometry within the block can only be edited to certain sizes specified in the block definition. When you use a grip to edit the block reference, tick marks are displayed at the locations of valid values for the block reference. If you change a block property value to a value other than one specified in the definition, the parameter will adjust to the closest valid value. For example, a block is defined to have a length of 2, 4, and 6. When you try to change that distance value to 10, the resulting value is 6 because it is the closest valid value.

To manipulate a dynamic block using custom grips

1. In a drawing, select a dynamic block reference.
2. Use the grips to stretch or change the block.

To manipulate a dynamic block using custom properties

1. In a drawing, select a dynamic block reference.
2. In the Properties palette, under Custom, change the required values.

To reset block references in a drawing

1 In a drawing, select a dynamic block reference.

2 Right-click in the drawing area. Click Reset Block.

NOTE To reset more than one block reference at a time, use the RESETBLOCK command.

Command entry: RESETBLOCK

To change the display color for custom grips

1 At the Command prompt, enter **gripdyncolor**.

2 Enter an integer from 1 to 255 (ACI color). Press Enter.

Quick Reference

Commands

ATTSYNC

Updates block references with new and changed attributes from a specified block definition.

INSERT

Inserts a block or drawing into the current drawing.

PROPERTIES

Controls properties of existing objects.

RESETBLOCK

Resets one or more dynamic block references to the default values of the block definition.

System Variables

BTMARKDISPLAY

Controls whether or not value set markers are displayed for dynamic block references.

GRIPDYNCOLOR

Controls the color of custom grips for dynamic blocks.

GRIPTIPS

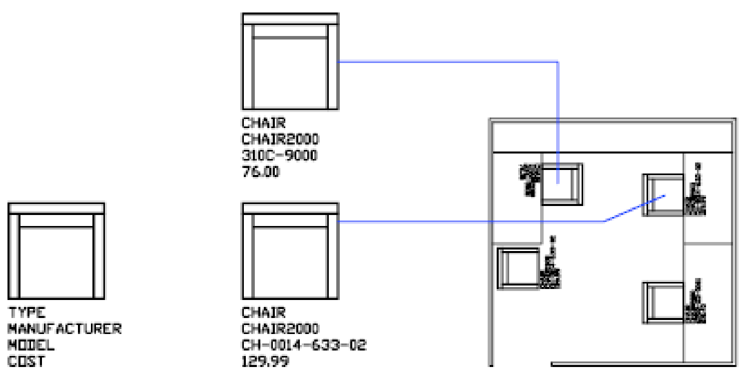
Controls the display of grip tips when the cursor hovers over grips on dynamic blocks and custom objects that support grip tips.

Attach Data to Blocks (Block Attributes)

You can attach information to blocks and later extract the information to create a bill of materials or other report.

Overview of Block Attributes

An attribute is a label or tag that attaches data to a block. Examples of data that might be contained in an attribute are part numbers, prices, comments, and owners' names. The tag is equivalent to a column name in a database table. The following illustration shows a block with four attributes: type, manufacturer, model, and cost.



The attributes in the illustration are single-line attributes. You can also create multiple-line attributes to store data such as addresses and descriptions.

Attribute information extracted from a drawing can be used in a spreadsheet or database to produce a parts list or a bill of materials. You can associate more than one attribute with a block, provided that each attribute has a different tag.

Attributes also can be "invisible." An invisible attribute is not displayed or plotted; however, the attribute information is stored in the drawing file and can be written to an extraction file for use in a database program.

Whenever you insert a block that has a variable attribute, you are prompted to enter data to be stored with the block. Blocks can also use constant attributes, attributes whose values do not change. Constant attributes do not prompt you for a value when you insert the block.

You can also create attributes. For more information about creating and working with an annotative attributes, see Create Annotative Blocks and Attributes on page 1417.

See also:

■ Modify a Block Attribute Definition on page 1049

■ Modify the Data in Block Attributes on page 1047

■ Scale Annotations on page 1393

Quick Reference

Commands

ATTDEF

Creates an attribute definition for storing data in a block.

ATTDISP

Controls the visibility overrides for all block attributes in a drawing.

ATTEDIT

Changes attribute information in a block.

ATTIPEDIT

Changes the textual content of an attribute within a block.

DDEDIT

Edits single-line text, dimension text, attribute definitions, and feature control frames.

PROPERTIES

Controls properties of existing objects.

System Variables

AFLAGS

Sets options for attributes.

ATTDIA

Controls whether the INSERT command uses a dialog box for attribute value entry.

ATTIPE

Controls the display of the in-place editor used to create multiline attributes.

ATTMODE

Controls display of attributes.

ATTMULTI

Controls whether multiline attributes can be created.

ATTREQ

Controls whether INSERT uses default attribute settings during insertion of blocks.

Define Block Attributes

The characteristics include the tag, which is a name that identifies the attribute, the prompt displayed when you insert the block, value information, text formatting, location within the block, and any optional modes (Invisible, Constant, Verify, Preset, Lock Position, and Multiple Lines).

If you plan to extract the attribute information for use in a parts list, you may want to keep a list of the attribute tags you have created. You will need this tag information later when you create the attribute template file.

Choose Attribute Modes

Attribute modes control the behavior of attributes in blocks. For example, you can control

- Whether an attribute is visible or invisible in the drawing
- Whether an attribute has a constant value, such as a part number
- Whether the attribute can be moved relative to the rest of the block

- Whether the attribute is a single-line attribute or a multiple-line attribute

If an attribute has a constant value, you will not be prompted for its value when you insert the block. If an attribute has a variable value, such as the asset number of a computer, you will be prompted when you insert the block.

Understand Single-Line and Multiple-Line Attributes

There are several differences between single-line and multiple-line attributes.

- Single-line attributes are limited to 255 characters from the user interface.
- Multiple-line attributes provide more formatting options than single-line attributes.
- When editing single-line and multiple line attributes, different editors are displayed.
- Multiple line attributes display four grips similar to MTEXT objects, while single-line attributes display only one grip.
- When a drawing is saved to AutoCAD 2007 or earlier, a multiple-line attribute is converted to several single-line attributes, one for every line of text in the original multiple-line attribute. If the drawing file is opened in the current release, these single line attributes are automatically merged back into a multiple-line attribute.

NOTE If a multiple-line attribute makes a round trip to an earlier release of AutoCAD, the differences between these two types of attributes might result in truncating very long lines of text and loss of formatting. However, before any characters are truncated, AutoCAD displays a dialog box that lets you cancel the operation.

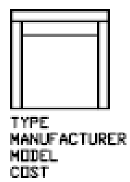
Correct Mistakes in Block Attribute Definitions

If you make a mistake, you can use the Properties palette or DDEDIT to make limited changes to an attribute definition before it is associated with a block. If you need to make more extensive changes, delete the attribute definition and create a new one.

Attach Attributes to Blocks

After you create one or more attribute definitions, you attach the attributes to a block when you define or redefine that block. When you are prompted to select the objects to include in the block definition, include in the selection set any attributes you want to attach to the block.

To use several attributes together, define them and then include them in the same block. For example, you can define attributes tagged "Type," "Manufacturer," “Model,” and “Cost,” and then include them in a block called CHAIR.



Usually, the order of the attribute prompts is the same as the order in which you selected the attributes when you created the block. However, if you used crossing or window selection to select the attributes, the order of the prompts is the reverse of the order in which you created attributes. You can use the Block Attribute Manager to change the order in which you are prompted for attribute information when you insert the block reference.

When you open a block definition in the Block Editor, you can use the Attribute Order dialog box to change the order in which you are prompted for attribute information when you insert the block reference.

Use Attributes Without Attaching Them to Blocks

Stand-alone attributes can also be created. Once attributes have been defined, and the drawing is saved, this drawing file can be inserted into another drawing. When the drawing is inserted, you are prompted for the attribute values.

To create an attribute definition

1. Click Home tab ➤ Block panel ➤ Define Attributes. 
2. In the Attribute Definition dialog box, set the attribute modes and enter tag information, location, and text options.
3. Click OK.

   After creating the attribute definition, you can select it as an object while creating a block definition. If the attribute definition is incorporated into a block, whenever you insert the block, you are prompted with the text string you specified for the attribute. Each subsequent instance of the block can have a different value specified for the attribute.

Command entry: ATTDEF

To create a multiline attribute definition

1 Click Home tab ➤ Block panel ➤ Define Attributes. 

2 In the Attribute Definition dialog box, under Mode, select Multiple Lines.

3 Enter tag information, location, and text options.

4 (Optional) Under Text Settings, Boundary Width, specify a value.

5 (Optional) Click the Multiline In-Place Text Editor button to use the in-place text editor to format the attribute in the drawing.

6 Click OK.

After creating the attribute definition, you can select it as an object while creating a block definition. If the attribute definition is incorporated into a block, whenever you insert the block, you are prompted with the text string you specified for the attribute. Each subsequent instance of the block can have a different value specified for the attribute.

Command entry: ATTDEF

To edit an attribute definition before it is associated with a block

1 Click Modify menu ➤ Object ➤ Text. 

2 Select the attribute to edit.

3 In the Edit Attribute Definition dialog box, specify the attribute tag, prompt, and default value. Then click OK.

Command entry: DDEDIT

To change the prompt order of attribute definitions

1 Click Insert tab ➤ Block panel ➤ Block Editor. 

2 In the Block Editor, select a block attribute.

3 Right-click in the Block Editor drawing area.

4 Click Attribute Order.

5 In the Attribute Order dialog box, select an attribute definition.

6 Click Move Up or Move Down to change the prompt order of the attribute definitions.

7 Repeat steps 2 and 3 until the attribute definition list is in the desired order.

8 Click OK.

Command entry: BEDIT ➤ BATTORDER

Quick Reference

Commands

ATTDEF

Creates an attribute definition for storing data in a block.

ATTDISP

Controls the visibility overrides for all block attributes in a drawing.

BATTORDER

Specifies the order of attributes for a block.

DDEDIT

Edits single-line text, dimension text, attribute definitions, and feature control frames.

PROPERTIES

Controls properties of existing objects.

System Variables

AFLAGS

Sets options for attributes.

ATTIPE

Controls the display of the in-place editor used to create multiline attributes.

ATTMULTI

Controls whether multiline attributes can be created.

Extract Data from Block Attributes

Extracting attribute information is an easy way to produce a schedule or bill of materials directly from your drawing data. For example, a facilities drawing might contain blocks representing office equipment. If each block has attributes identifying the model and manufacturer of the equipment, you can generate a report that estimates the cost of the equipment.

The Data Extraction wizard guides you through selecting drawings, block instances, and attributes. The wizard can also create a file with a *.dxe* file extension that contains all the settings for later reuse.

Output to a Table

If you extract attribute data to a table, the table is inserted in the current drawing and current space (model space or paper space) and on the current layer.

When you update the table, the attribute information is extracted again and the data rows in the table are replaced. If you have included a title row or one or more header rows in the table, they are not replaced during the update.

NOTE For access to shortcut menus in the drawing area that are needed for editing and updating tables, the Shortcut Menus in Drawing Area must be checked in the Options dialog box, User Preferences tab.

Output to a File

If you save the data to an external file, the comma-separated (CSV), tab-separated (TXT), Microsoft Excel (XLS), and Microsoft Access (MDB) file formats are available.

When the characters period (.), comma (,), or pound sign (#) are written to an Excel or Access file, they are replaced with their Unicode representation.

See also:

- Extract Data from Drawings and Spreadsheets

To extract block attributes to a table or a file

- Click Tools menu ➤ Data Extraction.
  The Data Extraction Wizard is displayed. The wizard provides step-by-step instructions for extracting information from block attributes in the current drawing or other drawings. The information is used to create a table in the current drawing or is saved to an external file.

  The Data Extraction wizard topic describes the options in the wizard.

Toolbar: Modify II 
Command entry: DATAEXTRACTION

To update extracted attribute data in a table

- When the Outdated Table dialog box is displayed as you open or save a drawing, click Update.

To turn off update notification for extracted attribute data in tables

1. At the Command prompt, enter **dxeval**.
2. Enter **0**.

Quick Reference

Commands

DATAEXTRACTION

Extracts drawing data and merges data from an external source to a data extraction table or external file.

System Variables

DXEVAL

Controls when update notification displays for data extraction tables.

Extract Block Attribute Data (Advanced)

You can extract attribute information from a drawing and create a separate text file for use with database software. This feature is useful for creating parts lists with information already entered in the drawing database. Extracting attribute information does not affect the drawing.

To create a parts list

- Create and edit an attribute definition
- Enter values for the attributes as you insert the blocks
- Create a template file and then extract attribute information to a text file

To extract attribute information, you first create an attribute template file using any text processor, then generate the attribute extraction file using AutoCAD, and, finally, open the attribute extraction file in a database application. If you plan to extract the attribute information to a DXF (drawing interchange format) file, it is not necessary to first create an attribute template file.

NOTE Make sure that the attribute extraction file does not have the same name as the attribute template file.

Create an Attribute Extraction Template File

Before you extract attribute information, you must create an ASCII template file to define the structure of the file that will contain the extracted attribute information. The template file contains information about the tag name, data type, field length, and number of decimal places associated with the information you want to extract.

Each field in the template file extracts information from the block references in the drawing. Each line in the template file specifies one field to be written to the attribute extraction file, including the name of the field, its character width, and its numerical precision. Each record in the attribute extraction file includes all the specified fields in the order given by the template file.

The following template file includes the 15 possible fields. *N* means numeric, *C* means character, *www* means a 3 digit number for the total width of the field, and *ddd* means a 3 digit number representing how many numeric decimal places are to be displayed to the right of the decimal point.

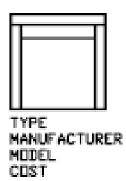
BL:NAME *Cwww*000 *(Block name)*
BL:LEVEL *Nwww*000 *(Block nesting level)*
BL:X *Nwwwddd(X coordinate of block insertion point)*
BL:Y *Nwwwddd(Y coordinate of block insertion point)*
BL:Z *Nwwwddd(Z coordinate of block insertion point)*
BL:NUMBER *Nwww*000 *(Block counter; the same for MINSERT)*

BL:HANDLE *Cwww*000 *(Block handle; the same for MINSERT)*
BL:LAYER *Cwww*000 *(Block insertion layer name)*
BL:ORIENT *Nwwwddd(Block rotation angle)*
BL:XSCALE *Nwwwddd(X scale factor)*
BL:YSCALE *Nwwwddd(Y scale factor)*
BL:ZSCALE *Nwwwddd(Z scale factor)*
BL:XEXTRUDE *Nwwwddd(X component of block extrusion direction)*
BL:YEXTRUDE *Nwwwddd(Y component of block extrusion direction)*
BL:ZEXTRUDE *Nwwwddd(Z component of block extrusion direction)*
numericNwwwddd (Numeric attribute tag)
*characterCwww*000 *(Character attribute tag)*

The template file can include any or all of the BL:*xxxxxxx* field names listed, but must include at least one attribute tag field. The attribute tag fields determine which attributes, hence which blocks, are included in the attribute extraction file. If a block contains some, but not all, of the specified attributes, the values for the absent ones are filled with blanks or zeros, depending on whether the field is a character field or a numeric field.

Comments should not be included in an attribute template file.

The illustration and table show an example of the type of information you're likely to extract, including block name, manufacturer, model number, and cost.



Field	**(C)haracter or (N)umeric data**	**Maximum field length**	**Decimal places**
Block name	C	040	000
Manufacturer	C	006	000
Model	C	015	000
Cost	N	006	002

You can create any number of template files, depending on how you'll use the data. Each line of a template file specifies one field to be written in the attribute extraction file.

Follow these additional guidelines:

- Be sure to place a space between the attribute tag and the character or numeric data. Use SPACEBAR, not TAB, to enter the space.
- Press Enter at the end of each line, including the last line.
- Each attribute extraction template file must include at least one attribute tag field, but the same field can appear only once in the file.

The following is a sample template file.

BL:NAME C008000 *(Block name, 8 characters)*

BL:X N007001 *(X coordinate, format nnnnnn.d)*

BL:Y N007001 *(Y coordinate, format nnnnnn.d)*

SUPPLIER C016000 (*Manufacturer's name, 16 characters)*

MODEL C009000 *(Model number, 9 characters)*

PRICE N009002 *(Unit price, format nnnnnnnn.dd)*

NOTE The format code for a numeric field includes the decimal point in the total field width. For example, the minimum field width to accommodate the number 249.95 would be 6 and would be represented as N006002. Character fields do not use the last three digits of the format code.

Create an Attribute Extraction File

After creating a template file, you can extract the attribute information using one of the following formats:

- Comma-delimited format (CDF)
- Space-delimited format (SDF)
- Drawing interchange format (DXF)

The CDF format produces a file containing one record for each block reference in a drawing. A comma separates the fields of each record, and single quotation marks enclose the character fields. Some database applications can read this format directly.

The SDF format also produces a file containing one record for each block reference in a drawing. The fields of each record have a fixed width and employ neither field separators nor character-string delimiters. The dBASE III Copy . . . SDF operation also produces SDI-format files. The Append From... SDF operation can read a file in dBASE IV format, which user programs written in FORTRAN can easily process.

DXF produces a subset of the drawing interchange format containing only block reference, attribute, and end-of-sequence objects. This option requires no attribute extraction template. The file extension *.dxx* distinguishes an extraction file in DXF format from normal DXF files.

Use the Attribute Extraction File

The attribute extraction file lists values and other information for the attribute tags you specified in the template file.

If you specified a CDF format using the sample template, the output might appear as follows:

'DESK', 120.0, 49.5, 'ACME INDUST.', '51-793W', 379.95
'CHAIR', 122.0, 47.0, 'ACME INDUST.', '34-902A', 199.95
'DESK', -77.2, 40.0, 'TOP DRAWER INC.', 'X-52-44',249.95

By default, character fields are enclosed with single quotes (apostrophes). The default field delimeter is a comma. The following two template records can be used to override these defaults:

C:QUOTE c (Character string delimeter)
C:DELIM c (Field delimeter)

The first nonblank character following the C:QUOTE or C:DELIM field name becomes the respective delimiter character. For example, if you want to enclose character strings with double quotes, include the following line in your attribute extraction template file:

C:QUOTE "

The quote delimiter must not be set to a character that can appear in a character field. Similarly, the field delimiter must not be set to a character that can appear in a numeric field.

If you specified an SDF format using the sample template, the file might be similar to the following example.

(NAME)	(X)	(Y)	(SUPPLIER)	(MODEL)	(PRICE)
DESK	120.0	49.5	ACME IN-DUST.	51-793W	379.95
CHAIR	122.0	47.0	ACME IN-DUST.	34-902A	199.95

(NAME)	(X)	(Y)	(SUPPLIER)	(MODEL)	(PRICE)
DESK	-77.2	40.0	TOP DRAWER INC.	X-52-44	249.95

The order of the fields corresponds to the order of the fields in the template files. You can use these files in other applications, such as spreadsheets, and you can sort and manipulate the data as needed. For example, you can open an attribute extraction file in Microsoft Excel in which you can specify a separate column for each field. See the documentation for your spreadsheet program for information about how to use data from other applications. If you open the file in Notepad or another Windows text processor, you can paste the information back into the drawing as text.

Nested Blocks

The line BL:LEVEL in a template file reports the nesting level of a block reference. A block that is inserted in a drawing has a nesting level of 1. A block reference that is part of (nested within) another block has a nesting level of 2, and so on.

For a nested block reference, the *X*,*Y*, *Z* coordinate values, scale factors, extrusion direction, and rotation angle reflect the actual location, size, orientation, and rotation of the nested block in the world coordinate system.

In some complex cases, nested block references cannot be correctly represented with only two scale factors and a rotation angle, for example, if a nested block reference is rotated in 3D. When this happens, the scale factors and rotation angle in the extracted file record are set to zero.

Error Handling

If a field is not wide enough for the data that is to be placed in it, the data is truncated and the following message is displayed:

** Field overflow in record <*record number*>

This could happen, for example, if you have a BL:NAME field with a width of 8 characters and a block in your drawing has a name 10 characters long.

To create an attribute extraction template file

1 Start Notepad.

   You can use any text editor or word processor that can save a text file in ASCII format.

2 Enter template information in Notepad. See Extract Block Attribute Data (Advanced) on page 1035 for format information.

3 Save the file with a *.txt* file extension.

   To extract data about a specific tag, insert the tag name in place of the "numeric" or "character" fields.

WARNING Do not use tab characters when constructing the template file with a word processor. If you use tab character alignment, the attribute information file is not created. To align the columns, insert ordinary spaces by pressing SPACEBAR. The use of tab characters may cause inconsistent alignment.

To extract attribute information

1 At the Command prompt, enter **attext**.

2 In the Attribute Extraction dialog box, specify the appropriate file format: CDF, SDF, or DXF.

3 Specify the objects to extract attributes from by choosing Select Objects.

   You can select a single block or multiple blocks in the drawing.

4 Specify the attribute template file to use by entering the file name or by choosing Template File and browsing.

5 Specify the output attribute information file by entering the file name or by choosing Output File and browsing.

6 Click OK.

Command entry: ATTEXT

Quick Reference

Commands

ATTEXT

Extracts attribute data, informational text associated with a block, into a file.

DATAEXTRACTION

Extracts drawing data and merges data from an external source to a data extraction table or external file.

Modify Blocks

You can modify a block definition or a block reference already inserted in the drawing.

Modify a Block Definition

You can redefine block definitions in your current drawing. Redefining a block definition affects both previous and future insertions of the block in the current drawing and any associated attributes.

There are two methods for redefining a block definition:

- Modify the block definition in the current drawing.
- Modify the block definition in the source drawing and reinsert it into the current drawing.

The method you choose depends on whether you want to make changes in the current drawing only or in a source drawing also.

Modify a Block Definition in the Current Drawing

To modify a block definition, follow the procedure to create a new block definition, but enter the name of the existing block definition. This replaces the existing block definition, and all the references to that block in the drawing are immediately updated to reflect the new definition.

To save time, you can insert and explode an instance of the original block and then use the resulting objects in creating the new block definition.

Update a Block Definition That Originated from a Drawing File

Block definitions created in your current drawing by inserting a drawing file are not updated automatically when the original drawing is modified. You can use INSERT to update a block definition from the drawing file.

Update a Block Definition That Originated from a Library Drawing (Advanced)

DesignCenter™ does not overwrite an existing block definition in a drawing with one that comes from another drawing. To update a block definition that came from a library drawing, use WBLOCK to create a separate drawing file from the library drawing block. Then, use INSERT to overwrite the block definition in the drawing that uses the block.

NOTE Block descriptions are stripped off when using INSERT. Use the Clipboard to copy and paste a block description displayed in the Block Definition dialog box from one block definition to another.

Modify the Description of a Block

To modify the DesignCenter description of a block definition, use BLOCK. You can also add descriptions to any number of existing blocks in the Block Definition dialog box.

Redefine Block Attributes

You can attach attributes to a block when you define or redefine that block. When you are prompted to select the objects to include in the block definition, include the desired attributes in the selection set. Redefining the attributes in the block definition has the following effects on block references that were previously inserted:

- Constant attributes, which have a fixed value, are lost and replaced by any new constant attributes.
- Variable attributes remain unchanged, even if the new block definition has no attributes.
- New attributes do not appear in the existing block references.

See also:

- Attach Data to Blocks (Block Attributes) on page 1027
- Modify Dynamic Block Definitions on page 888

To update a block definition that originated from a drawing file

1. If DesignCenter is not already open, click Tools menu ➤ Palettes ➤ DesignCenter.
2. In the tree view, click the folder that contains the drawing file from which the block originated.
3. In the content area (on the right side), right-click the drawing file.
4. On the shortcut menu, click Insert as Block.
5. In the Insert dialog box, click OK.
6. In the Block - Redefine Block dialog box, click Redefine Block.

7 Press ESC to exit the command.

To modify a block description

1 Click Modify menu ➤ Object ➤ Block Description.

2 In the Block Definition dialog box, in the Name list, select the block for which you want to modify the block description.

3 In the Description box, enter or modify the description of the block.

4 Click OK.

5 In the Block - Redefine Block dialog box, click Redefine Block.

Command entry: BLOCK

Quick Reference

Commands

BLOCK

Creates a block definition from selected objects.

EXPLODE

Breaks a compound object into its component objects.

PROPERTIES

Controls properties of existing objects.

Change the Color and Linetype in a Block

Blocks containing objects with floating properties inherit their color and linetype from the layer on which they are inserted. Depending on how the objects in the block were created, blocks can also inherit floating color and linetype properties from the current explicit color and linetype that you set to override the layer settings.

If a block was not created using objects with floating color and linetype properties, the only way to change these properties is to redefine the block.

See also:

- Control the Color and Linetype Properties in Blocks on page 860

To change the layer of an object

1. Select the objects whose layer you want to change.
2. On the Layers toolbar, click the Layer control.
3. Select the layer that you want to assign to the objects.

Command entry: LAYER

To change the color assigned to a layer

1. On the Layers toolbar, click the Layer Properties Manager button.
2. In the Layer Properties Manager, click the color you want to change.
3. In the Select Color dialog box, use one of the following methods:
   - On the Index tab, click a color and then click OK.
   - On the Index tab, enter the ACI color number (1-255) or name in the Color box, and then click OK.
   - On the True Color tab, select the HSL color model in the Color Model option and specify a color by entering a color value in the Color box or by specifying values in the Hue, Saturation, and Luminance boxes, and then click OK.
   - On the Color Books tab, select a color book from the Color Book box, select a color by navigating the color book (using the up and down arrows) and clicking on a color chip, and then click OK.
4. Click OK.

Toolbar: Layers 
Command entry: LAYER

To change the color of an object, overriding the layer's color

1. On the Standard toolbar, click Properties.
2. Select the objects whose color you want to change.

3 In the Properties palette, select Color.

   An arrow is displayed in the right column.

4 Click the arrow and select a color from the list.

Toolbar: Standard 
Command entry: PROPERTIES

To change the linetype assigned to a layer

1 On the Layers toolbar, click the Layer Properties Manager button.

2 In the Layer Properties Manager, click Load and click one or more linetypes to load, and then click OK.

   You can hold down Ctrl to select several linetypes or Shift to select a range of linetypes.

3 Select a layer from the list and then click Details to expand the dialog box.

4 Select a linetype from the Linetype list.

5 Click OK to close the dialog box.

   To open the Layer Properties Manager, click Format menu ➤ Layer.

Toolbar: Layers 
Command entry: LAYER

To change the linetype of an object, overriding the layer's linetype

1 Select the objects whose linetype you want to change.

2 On the Properties toolbar, click the Linetype control.

3 Click the linetype that you want to assign to the objects.

Command entry: LINETYPE

To modify an existing block definition

1 Select the block to modify.

2 Right-click the block and click Properties on the shortcut menu.

3 In the Properties palette, select and modify *X* and *Y* position, scale, rotation values, or other properties.

Command entry: PROPERTIES

Quick Reference

Commands

LAYER

Manages layers and layer properties.

LINETYPE

Loads, sets, and modifies linetypes.

PROPERTIES

Controls properties of existing objects.

Modify the Data in Block Attributes

You can use any of the following methods to edit the values of attributes attached to a block:

- Double-click the block to display the Enhanced Attributes Editor
- Press Ctrl and double-click the attribute to display the in-place editor
- Open the Properties palette and select the block
- Enter the ATTEDIT command to display the Edit Attributes dialog box
- Enter the -ATTEDIT command for Command prompt access to multiple attribute values and properties

You can also change the location of attributes in a block using grips. With multiple-line attributes, you can also move grips to resize the width of the text.

NOTE If you press Ctrl and double-click an attribute that includes a hyperlink, the hyperlink opens the web page. To edit the attribute, use one of the other methods listed.

See also:

- Modify a Block Definition on page 1042

To edit attribute data

1. Click Modify menu ➤ Objects ➤ Attribute ➤ Single. 
2. Select the block to edit.
3. In the Edit Attributes dialog box, retype the attribute information as necessary and click OK.

Quick Reference

Commands

ATTEDIT

Changes attribute information in a block.

ATTIPEDIT

Changes the textual content of an attribute within a block.

ATTSYNC

Updates block references with new and changed attributes from a specified block definition.

BATTMAN

Manages the attributes for a selected block definition.

EATTEDIT

Edits attributes in a block reference.

System Variables

ATTIPE

Controls the display of the in-place editor used to create multiline attributes.

ATTMULTI

Controls whether multiline attributes can be created.

Modify a Block Attribute Definition

You can modify attributes in block definitions with the Block Attribute Manager. For example, you can modify the following:

- Properties that define how values are assigned to an attribute and whether or not the assigned value is visible in the drawing area
- Properties that define how attribute text is displayed in the drawing
- Properties that define the layer that the attribute is on and the attribute line's color, weight, and type

By default, attribute changes you make are applied to all existing block references in the current drawing.

Changing the attribute properties of existing block references does not affect the values assigned to those blocks. For example, in a block containing an attribute whose tag is Cost and value is 19.99, the 19.99 value is unaffected if you change the tag from Cost to Unit Cost.

Updating attributes with duplicate tag names can lead to unpredictable results. Use the Block Attribute Manager to find duplicate tags and change tag names.

If constant attributes or nested attributed blocks are affected by your changes, use REGEN to update the display of those blocks in the drawing area.

Change the Prompt Order for Attribute Values

When you define a block, the order in which you select the attributes determines the order in which you are prompted for attribute information when you insert the block. You can use the Block Attribute Manager to change the order of prompts that request attribute values.

Remove Block Attributes

You can remove attributes from block definitions and from all existing block references in the current drawing. Attributes removed from existing block references do not disappear in the drawing area until you regenerate the drawing using REGEN.

You cannot remove all attributes from a block; at least one attribute must remain. If you need to remove all attributes, redefine the block.

Update Block References

You can update attributes in all block references in the current drawing with changes you made to the block definition. For example, you may have used the Block Attribute Manager to modify attribute properties in several block definitions in your drawing but elected not to automatically update existing block references when you made the changes. Now that you are satisfied with the attribute changes you made, you can apply those changes to all blocks in the current drawing.

You can also use ATTSYNC to update attribute properties in block references to match their block definition, or to update a block instance after you redefine a block attribute using BLOCK, -BLOCK, or BEDIT.

Updating attribute properties in block references does not affect any values that have been assigned to those attributes.

Edit Attributes in a Block Reference

You can select an attribute in a block reference and use the Properties palette to change its properties, or you can use the Enhanced Attribute Editor to modify all the attributes in a selected block reference.

See also:

- Define Block Attributes on page 1029
- Modify a Block Definition on page 1042

To edit attributes assigned to a block definition

1. Click Home tab ➤ Block panel ➤ Manage Attributes.
2. In the Block Attribute Manager, select a block from the Block list, or click Select Block and select a block in the drawing area.
3. In the list of attributes, double-click the attribute you want to edit, or select the attribute and click Edit.
4. In the Edit Attribute dialog box, make the attribute changes you want, and then click OK.

Toolbar: Modify II

Command entry: BATTMAN

To specify whether changes are applied to existing block references

1 Click Home tab ➤ Block panel ➤ Manage Attributes.

2 In the Block Attribute Manager, click Settings.

3 In the Settings dialog box, do one of the following:
   - To apply changes to existing block references, select the Apply Changes to Existing References option.
   - To apply changes only to new block insertions, clear the Apply Changes to Existing References option.

4 Click OK.

Toolbar: Modify II
Command entry: BATTMAN

To highlight duplicate attribute tags in a block

1 Click Home tab ➤ Block panel ➤ Manage Attributes.

2 In the Block Attribute Manager, click Settings.

3 In the Settings dialog box, select Emphasize Duplicate Tags.

4 Click OK.

Toolbar: Modify II
Command entry: BATTMAN

To change the prompt order for attribute values

1 Click Home tab ➤ Block panel ➤ Manage Attributes.

2. In the Block Attribute Manager, select a block from the Block list, or click Select Block and select a block in the drawing area.

   For the selected block, attributes are listed in their prompt order.

3. To move an attribute up in the prompt order, select the attribute, and then click Move Up; to move an attribute down in the prompt order, select the attribute, and then click Move Down.

   NOTE The Move Up and Move Down buttons are unavailable for attributes with constant values (Mode=C).

Toolbar: Modify II
Command entry: BATTMAN

To remove an attribute from a block definition and all block references

1. Click Home tab ➤ Block panel ➤ Manage Attributes.
2. In the Block Attribute Manager, select a block from the Block list, or click Select Block and select a block in the drawing area.
3. (Optional) If you do not want attributes removed from existing instances of the block, click Settings, and, in the Settings dialog box, clear Apply Changes to Existing References.
4. In the Block Attribute Manager, select an attribute from the attribute list, and then click Remove.

   Attributes removed from existing block instances do not disappear until you regenerate the drawing using REGEN.

Toolbar: Modify II
Command entry: BATTMAN

To update existing block references with attributes you have modified

1. Click Home tab ➤ Block panel ➤ Manage Attributes.

2 In the Block Attribute Manager, select a block from the Block list, or click Select Block and select a block in the drawing area.

3 Click Sync to update attributes you have changed in all block references for the selected block.

Toolbar: Modify II 
Command entry: BATTMAN

To update the attributes in block references for a selected block definition

1 Click Home tab ➤ Block panel ➤ Synchronize Attributes. 

2 At the prompt, do one of the following:
   - Enter **name**, and then enter the name of the block whose block references you want to update.
   - Enter **?** to view a list of blocks, and then enter **name**, followed by the name of the block.
   - Press Enter, and then use your pointing device to select a block in the drawing area.

   If you specify a block that does not exist, or if the block exists but does not contain attributes, an error message is displayed.

Toolbar: Modify II 
Command entry: ATTSYNC

Alternate

1 Click Home tab ➤ Block panel ➤ Edit Single Attribute. 

2 In the drawing area, select the block you want to edit.

3 In the Enhanced Attribute Editor, select the attribute you want to edit. You can change the attribute value or choose another tab and edit other attribute properties.

4 Make the attribute changes you want, and then do one of the following:

- Click Apply to save your changes. The Enhanced Attribute Editor remains open.
  If you click Cancel later to exit the Enhanced Attribute Editor, attribute changes you made prior to choosing Apply are not reversed.
- Click OK to save your changes and close the Enhanced Attribute Editor.
- Click Select Block to edit the attributes of a different block. If you made changes to the current block, but have not saved them, you are prompted to do so before selecting a new block.

Toolbar: Modify II 
Command entry: EATTEDIT

Quick Reference

Commands

ATTIPEDIT

Changes the textual content of an attribute within a block.

ATTSYNC

Updates block references with new and changed attributes from a specified block definition.

BATTMAN

Manages the attributes for a selected block definition.

EATTEDIT

Edits attributes in a block reference.

System Variables

ATTIPE

Controls the display of the in-place editor used to create multiline attributes.

ATTMULTI

Controls whether multiline attributes can be created.

Disassemble a Block Reference (Explode)

If you need to modify one or more objects within a block separately, you can disassemble, or explode, the block reference into its component objects. After making the changes, you can

- Create a new block definition
- Redefine the existing block definition
- Leave the component objects uncombined for other uses

You can automatically explode block references as you insert them by selecting the Explode option in the Insert dialog box.

To explode a block reference

1. Click Home tab ➤ Modify panel ➤ Explode. 
2. Select the block to explode. Press Enter.

   The block reference is disassembled into its component objects; however, the block definition still exists in the drawing for insertion later.

Toolbar: Modify 
Command entry: EXPLODE

To control properties while you explode an object

1. Enter **xplode**.
2. Select the objects to be exploded.
3. If you selected more than one object, enter **i** to control properties for individual objects, or enter **g** to control properties for all the selected objects.
4. Enter an option for a property you want to change.

   The property is applied to the compound object and the prompt is redisplayed.
5. Enter another option, or enter **e** to explode the selected objects.

The selected objects are exploded and the properties you specified are applied to the component objects.

Quick Reference

Commands

EXPLODE

Breaks a compound object into its component objects.

XPLODE

Breaks a compound object into its component objects.

Change Existing Objects

19

You can select objects, view and edit object properties, and perform general and object-specific editing operations.

Select Objects

You have a wide range of options when you need to select objects for editing operations.

Select Objects Individually

At the Select Objects prompt, you can select one or more objects individually.

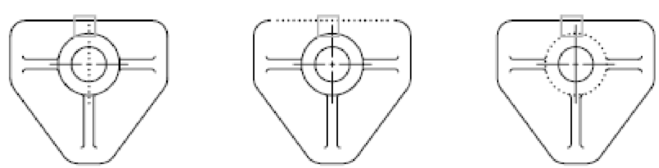
Use the Pickbox Cursor

When the square pickbox cursor is in position to select an object, the object is highlighted. Click to select the object.

You can control the size of the pickbox in the Options dialog box, Selection tab.

Select Objects Close Together

It is difficult to select objects that are close together or lie directly on top of one another. The example shows two lines and a circle that lie within the pickbox.



first selected object   second selected object   third selected object

If selection preview is turned on, you can cycle through the objects by rolling over the object on top to highlight it, and pressing and holding Shift and then pressing Spacebar continuously. When the required object is highlighted, left-click to select it.

If selection preview is turned off, hold down Shift + Spacebar and click to cycle through these objects, one after the other, until the one you want is selected. Press Esc to turn off cycling.

Remove Selection from Objects

Remove objects from the current selection set by holding down Shift and selecting them again.

See also:

- Modify 3D Solid Subobjects on page 1310
- Modify Composite Solids and Surfaces on page 1328

To select a single object

1. At the Select Objects prompt of any command, move the rectangular pickbox cursor so that the object that you want to select is highlighted.
2. Click the object.

   The object you selected is highlighted.
3. Press Enter to end object selection.

   NOTE If the PICKFIRST system variable is set to 1 (noun-verb selection), you can select objects before entering a command.

To change the size of the pickbox cursor

1. Click Tools menu ➤ Options.
2. On the Selection tab, under Pickbox Size, move the pickbox size slider until the pickbox is the size you want to use.
3. Click OK.

Command entry: PICKBOX

To cycle through objects for selection

1. At the Select Objects prompt, hold down Shift + Spacebar. Click as near as possible to the object you want.
2. Keep clicking until the object you want is highlighted.
3. Press Enter to select the object.

NOTE If selection preview is turned on, you can cycle through the objects by rolling over the object on top to highlight it, and pressing and holding Shift and then pressing Spacebar continuously. When the required object is highlighted, left-click to select it.

To remove selection from objects

- Hold down Shift. Click the objects that you want removed from the selection set.

Quick Reference

Commands

PROPERTIES

Controls properties of existing objects.

SELECT

Places selected objects in the Previous selection set.

System Variables

3DSELECTIONMODE

Controls the selection precedence of visually overlapping objects when using 3D visual styles.

HIGHLIGHT

Controls object highlighting; does not affect objects selected with grips.

LEGACYCTRLPICK

Specifies the keys for selection cycling and the behavior for Ctrl+click.

PICKADD

Controls whether subsequent selections replace the current selection set or add to it.

PICKAUTO

Controls automatic windowing at the Select Objects prompt.

PICKBOX

Sets the object selection target height, in pixels.

PICKDRAG

Controls the method of drawing a selection window.

PICKFIRST

Controls whether you select objects before (noun-verb selection) or after you issue a command.

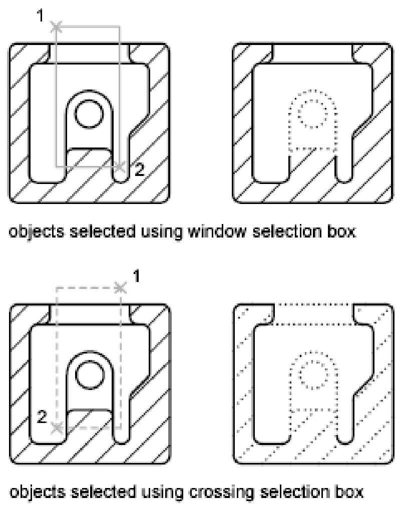
Select Multiple Objects

At the Select Objects prompt, you can select many objects at the same time.

Specify a Rectangular Selection Area

Specify opposite corners to define a rectangular area. The background inside the area changes color and becomes transparent. The direction that you drag your cursor from the first point to the opposite corner determines which objects are selected.

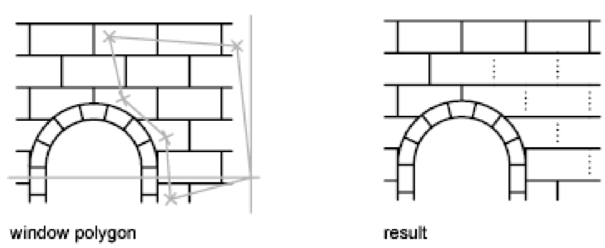
- **Window selection.** Drag your cursor from left to right to select only objects that are entirely enclosed by the rectangular area.
- **Crossing selection.** Drag your cursor from right to left to select objects that the rectangular window encloses or crosses.



With a window selection, usually the entire object must be contained in the rectangular selection area. However, if an object with a noncontinuous (dashed) linetype is only partially visible in the viewport and all the visible vectors of the linetype can be enclosed within the selection window, the entire object is selected.

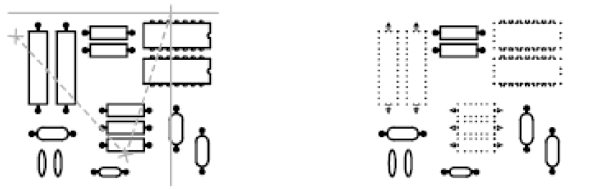
Specify an Irregularly Shaped Selection Area

Specify points to define an irregularly shaped area. Use window polygon selection to select objects entirely enclosed by the selection area. Use crossing polygon selection to select objects enclosed *or* crossed by the selection area.



Specify a Selection Fence

In a complex drawing, use a selection fence. A selection fence looks like a polyline and selects only the objects it passes through. The circuit board illustration shows a fence selecting several components.



fence selection   selected objects highlighted

Use Other Selection Options

You can see all selection options by entering **?** at the Select Objects prompt. For a description of each of the selection options, see *SELECT*.

Remove Selection from Multiple Objects

You can enter **r** (Remove) at the Select Objects prompt and use any selection option to remove objects from the selection set. If you are using the Remove option and want to return to adding objects to the selection set, enter **a** (Add).

You can also remove objects from the current selection set by holding down Shift and selecting them again, or by holding down Shift and then clicking and dragging window or crossing selections. You can add and remove objects repeatedly from the selection set.

To see a list of options at the Select Objects prompt

- Enter **?** at the Select Objects prompt.

To select objects within an irregularly shaped area

1. At the Select Objects prompt, enter **wp** (Window Polygon).
2. Specify points that define an area entirely enclosing the objects you want to select.
3. Press Enter to close the polygon selection area and complete the selection.

To select objects crossing an irregularly shaped area

1. At the Select Objects prompt, enter **cp** (Crossing Polygon).
2. Specify points that define an area that encloses or crosses the objects you want to select.
3. Press Enter to close the polygon selection area and complete the selection.

To select objects with a fence

1. At the Select Objects prompt, enter **f** (Fence).
2. Specify points to create a fence that passes through the objects you want to select.
3. Press Enter to complete the selection.

To remove several objects from the selection set

1. After selecting objects, at the Select Objects prompt, enter **r** (Remove).
2. Enter any selection option such as **cp** (Crossing Polygon) or **f** (Fence), and select the objects to be removed from the selection set.

   To return to adding objects to the selection set, enter **a** (Add).

Quick Reference

Commands

PROPERTIES

Controls properties of existing objects.

QSELECT

Creates a selection set based on filtering criteria.

SELECT

Places selected objects in the Previous selection set.

System Variables

HIGHLIGHT

Controls object highlighting; does not affect objects selected with grips.

PICKADD

Controls whether subsequent selections replace the current selection set or add to it.

PICKAUTO

Controls automatic windowing at the Select Objects prompt.

PICKBOX

Sets the object selection target height, in pixels.

PICKDRAG

Controls the method of drawing a selection window.

PICKFIRST

Controls whether you select objects before (noun-verb selection) or after you issue a command.

PREVIEWEFFECT

Specifies the visual effect used for previewing selection of objects.

Prevent Objects from Being Selected

You can prevent objects on specified layers from being selected and modified by locking those layers.

Typically, you lock layers to prevent accidental editing of particular objects. Other operations are still possible when a layer is locked. For example, you can make a locked layer current, and you can add objects to it. You can also use inquiry commands (such as LIST), use object snaps to specify points on objects on locked layers, and change the draw order of objects on locked layers.

To help you differentiate between locked and unlocked layers, you can do the following:

- Hover over an object to see whether a lock icon is displayed
- Dim the objects on locked layers

NOTE Grips are not displayed on objects that are on locked layers.

To lock or unlock a layer

1 Click Home tab ➤ Layers panel ➤ Layer Properties. 

2 In the Layer Properties Manager, click the padlock for the layers that you want to lock.

3 Click OK.

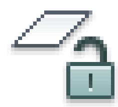
If the padlock is closed, the layer is locked and objects on that layer cannot be selected.

Toolbar: Layers 
Command entry: LAYER

To lock or unlock a layer by selecting an object on that layer

1 Click the Model tab.

2 Do either of the following:

- Click Home tab ➤ Layers panel ➤ Lock. 
- Click Home tab ➤ Layers panel ➤ Unlock. 

3 Select an object on the layer you want to lock or unlock.

Toolbar: Layer II
Command entry: LAYLCK,LAYULK

Quick Reference

Commands

LAYER

Manages layers and layer properties.

LAYISO

Hides or locks all layers except those of the selected objects.

LAYLCK

Locks the layer of a selected object.

LAYULK

Unlocks the layer of a selected object.

System Variables

LAYLOCKFADECTL

Controls the amount of fading for objects on locked layers.

Filter Selection Sets

You can use object properties or object types to include objects in a selection set, or to exclude them.

Using either Quick Select (QSELECT) from the Properties palette or the Object Selection Filters dialog box (FILTER), you can filter selection sets by property (such as color) and by object type. For example, you can select all of the red circles in a drawing without selecting any other object, or you can select all objects except the red circles.

With Quick Select, you can quickly define a selection set based on filtering criteria that you specify, and if an Autodesk or a third-party application was used to add a feature classification to an object, you can select objects by classification property. With object selection filters, you can name and save filters for future use.

With either Quick Select or object selection filters, if you want to filter your selection set based on color, linetype, or lineweight, first consider whether these properties are set to BYLAYER for any objects in your drawing. For example, an object may appear red because its color is set to BYLAYER and the layer color is red.

See also:

- Customize Object Selection on page 1070
- Work with Layers on page 533

To create a selection set using Quick Select

In the following example, you use Quick Select to select the red objects in a drawing.

1 Click Home tab ➤ Utilities panel ➤ Quick Select. 

2. In the Quick Select dialog box, under Apply To, select Entire Drawing.
3. Under Object Type, select Multiple.
4. Under Properties, select Color.
5. Under Operator, select Equals.
6. Under Value, select Red.
7. Under How to Apply, select Include in New Selection Set.
8. Click OK.

   All red objects in the drawing are selected and the Quick Select dialog box closes. Objects that are set to BYLAYER and are red because the layer color is red are not included in the selection set.

NOTE If an application such as Autodesk Map was used to add a feature classification to an object, and the associated classification (XML) file is present, you can select objects by classification property. Specifically, you can select a classification in the Object Type box and a property in the Properties box.

Command entry: QSELECT

To exclude objects from the selection set

You can exclude objects from the current selection set by using the Exclude from New Selection Set option. In the following example, you exclude all circles with a radius greater than 1 from a set of objects already selected.

1. Select several objects.
2. Click Home tab ➤ Utilities panel ➤ Quick Select. 
3. In the Quick Select dialog box, under Apply To, select Current Selection.
4. Under Object Type, select Circle.
5. Under Properties, select Radius.
6. Under Operator, select Greater Than.
7. Under Value, enter **1**.
8. Under How to Apply, select Exclude from New Selection Set.
9. Click OK.

All circles with a radius greater than 1 are removed from the selection set.

Command entry: QSELECT

To append objects to the selection set

You can use Quick Select to append objects to a current selection set. In the following example, you keep the current selection set and append all objects in the drawing that contain hyperlinks whose names begin with **bld1_**.

1 Click Home tab ➤ Utilities panel ➤ Quick Select.

2 In the Quick Select dialog box, select Append to Current Selection Set.

3 In the Object Type box, select Multiple.

4 Under Properties, select Hyperlink.

5 Under Operator, select Wildcard Match.

6 Under Value, enter **bld1_***.

7 Under How to Apply, select Include in New Selection Set.

8 Click OK.

Command entry: QSELECT

To name and save a filtered list

1 At the Command prompt, enter **filter**.

2 In the Object Selection Filters dialog box, under Select Filter, select a filter such as **Line**.

3 Click Add to List.

4 Under Save As, enter a filter name such as **Linefilter**.

5 Click Save As.

6 Click Apply.

   The filter is applied so you can select, in this case, only lines in the drawing. If you select objects with a selection, the filter is applied to all objects in the selection area.

Command entry: FILTER

To use a named filter

1. At the Select Object prompt, enter '**filter**. (The apostrophe makes it a transparent command.)
2. In the Object Selection Filters dialog box, under Select Filter, select the filter you want to use. Click Apply.
3. Use a crossing window to specify objects for selection.

   Only the objects selected by the crossing window that match the filter criteria are selected.

Quick Reference

Commands

FILTER

Creates a list of requirements that an object must meet to be included in a selection set.

PROPERTIES

Controls properties of existing objects.

QSELECT

Creates a selection set based on filtering criteria.

SELECT

Places selected objects in the Previous selection set.

System Variables

PICKADD

Controls whether subsequent selections replace the current selection set or add to it.

PICKAUTO

Controls automatic windowing at the Select Objects prompt.

PICKBOX

Sets the object selection target height, in pixels.

PICKDRAG

Controls the method of drawing a selection window.

PICKFIRST

Controls whether you select objects before (noun-verb selection) or after you issue a command.

Customize Object Selection

You can control several aspects of selecting objects, such as whether you enter a command first or select objects first, the size of the pickbox cursor, and how selected objects are displayed.

For commands that use the Select Objects prompt, you can

- Enter a command first, and then select objects
- Select the objects first, and then enter a command

You can also choose

- Whether objects to be selected are previewed during selection
- Whether selected objects are highlighted
- How you define selection areas and how you create selection sets

Select the Command First

When you use an editing command, a Select Objects prompt is displayed and the crosshairs is replaced with a pickbox. You can respond to the Select Objects prompt in various ways:

- Select objects one at a time.
- Click an empty area. Drag the cursor to define a rectangular selection area.
- Enter a selection option. Enter **?** to display all selection options.
- Combine selection methods. For example, to select most of the objects in the drawing area, select all objects and then remove the objects that you do not want selected.
- Enter '**filter** to use a named selection filter. The apostrophe runs the command transparently.

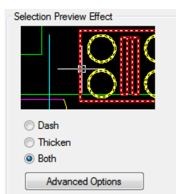
Select Objects First

You can use one of two methods to select objects before starting a command:

- Use the *SELECT* command, and enter **?** to display all selection options. All objects selected are put into the Previous selection set. To use the Previous selection set, enter **p** at the Select Objects prompt of any subsequent command.
- When noun/verb selection is turned on, select objects at the Command prompt before entering a command such as MOVE, COPY, or ERASE. With this method, you can only select objects by clicking them individually or by using automatic selection.
- Enter **qselect** to filter the selection. Then enter **p** at the Select Objects prompt of any subsequent command.

Highlight Objects to Be Selected

Objects are highlighted when the pickbox cursor rolls over them, providing a preview of which object will be selected when you click.



When you specify an area to select multiple objects, the background of the area becomes transparent.

These selection previewing effects are turned on by default. You can turn them off or change the appearance of selection previewing (Options dialog box, Selection tab). When the PICKBOX system variable is set to 0, selection previewing of objects is not available.

Control the Appearance of Selected Objects

By default, selected objects are displayed with dashed lines. You can increase program speed by setting the HIGHLIGHT system variable to 0. Turning off selection highlighting does *not* affect grips on selected objects.

Set Up Default Selection Methods

Options on the Selection tab of the Options dialog box control default selection methods:

- Use selection previewing and selection area effects to preview selection.
- Select objects before entering a command (noun-verb selection) or after entering a command. (PICKFIRST)
- Press Shift to append objects to the selection set. (PICKADD)
- Click and drag to create a selection window. Otherwise you must click twice to define the corners of a selection window. (PICKDRAG)
- Start Window or Crossing selection automatically when you click an empty space. Otherwise, you must enter **c** or **w** to specify window crossing selection. (PICKAUTO)
- Change the size of the pickbox. (PICKBOX)
- Select all objects in a group when you select one object in that group.
- Include the boundary in the selection set when you select a hatch.

To change the size of the pickbox cursor

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Selection tab, under Pickbox Size, move the slider until the pickbox is the size you want to use.
3. Click OK.

Command entry: OPTIONS

To change object selection settings

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Selection tab, make changes to the Selection Preview and Selection Modes areas and the pickbox size.
3. Click OK.

Command entry: OPTIONS

To turn on or turn off selection previewing

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Selection tab, select or clear options as follows:
   - Select the When a Command Is Active option to display the check mark.
   - Select the When No Command Is Active option to display the check mark.
   - Select both options to turn on selection preview whenever it is available.
   - Clear both options to turn off selection preview entirely.

   SELECTIONPREVIEW

To change the appearance of selection previewing

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Selection tab, click Visual Effect Settings.
3. In the Visual Effect Settings dialog box, select one of the following options:
   - **Dash**. Displays dashed lines.
   - **Thicken**. Displays thickened lines.
   - **Both**. Displays dashed and thickened lines.
4. Click OK to exit each dialog box.

To exclude objects from selection previewing

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Selection tab, click Visual Effect Settings.

3 In the Visual Effect Settings dialog box, click Advanced Options.

4 In the Advanced Preview Options dialog box, select any of the following options to exclude objects from selection previewing:

- Exclude Objects on Locked Layers
- Xrefs
- Tables
- Groups
- Multiline Text
- Hatches

5 Click OK to exit each dialog box.

To change the appearance of the selection area

1 Click Tools menu ➤ Options. 

2 In the Options dialog box, Selection tab, click Visual Effect Settings.

3 In the Visual Effect Settings dialog box, change any of the following settings:

- **Indicate Selection Area**. Select to display effects for selection areas.
- **Window Selection Color**. Select a color, or click Select Color to display the Select Color dialog box. (WINDOWAREACOLOR system variable)
- **Crossing Selection Color**. Select a color, or click Select Color to display the Select Color dialog box. (CROSSINGAREACOLOR system variable)
- **Selection Area Opacity**. Use the slider to set transparency for selection areas. The lower the setting, the more transparent the area. A value of 100 makes the area opaque. (SELECTIONAREAOPACITY system variable)

4 Click OK to exit each dialog box.

Quick Reference

Commands

FILTER

Creates a list of requirements that an object must meet to be included in a selection set.

PROPERTIES

Controls properties of existing objects.

QSELECT

Creates a selection set based on filtering criteria.

System Variables

CROSSINGAREACOLOR

Controls the color of the selection area during crossing selection.

DRAGMODE

Controls the way dragged objects are displayed.

HIGHLIGHT

Controls object highlighting; does not affect objects selected with grips.

PICKADD

Controls whether subsequent selections replace the current selection set or add to it.

PICKAUTO

Controls automatic windowing at the Select Objects prompt.

PICKBOX

Sets the object selection target height, in pixels.

PICKDRAG

Controls the method of drawing a selection window.

PICKFIRST

Controls whether you select objects before (noun-verb selection) or after you issue a command.

PREVIEWEFFECT

Specifies the visual effect used for previewing selection of objects.

PREVIEWFILTER

Excludes specified object types from selection previewing.

SELECTIONAREA

Controls the display of effects for selection areas.

SELECTIONAREAOPACITY

Controls the transparency of the selection area during window and crossing selection.

SELECTIONPREVIEW

Controls the display of selection previewing.

WINDOWAREACOLOR

Controls the color of the transparent selection area during window selection.

Group Objects

A group is a saved set of objects that you can select and edit together or separately as needed. Groups provide an easy way to combine drawing elements that you need to manipulate as a unit.

Overview of Groups

A group is a saved set of objects that you can select and edit together or separately as needed. Groups provide an easy way to combine drawing elements that you need to manipulate as a unit. You can create them quickly and with a default name.

TIP Groups are useful in associating 3D solids when you do not want to combine them with a Boolean operation.

You can change the components of groups as you work by adding or removing objects.

In some ways, groups resemble blocks, which provide another method of combining objects into a named set. For example, the groups you create are saved from session to session. However, you can edit individual objects in groups more easily than you can edit them in blocks, which must be exploded first. Unlike blocks, groups cannot be shared with other drawings.

Quick Reference

Commands

GROUP

Creates and manages saved sets of objects called groups.

System Variables

PICKSTYLE

Controls the use of group selection and associative hatch selection.

Create Groups

In addition to choosing the objects that will become the members of a group, you can give the group a name and description.

When you create a group, you can give the group a name and description. If you copy a group, the copy is given the default name Ax and is considered unnamed. Unnamed groups are not listed in the Object Grouping dialog box unless you select Include Unnamed.

If you choose a member of a group that can be selected for inclusion in a new group, all members of the former group are included in the new group.

The objects in your drawing can be members of more than one group, and groups themselves can be nested in other groups. You can ungroup a nested group to restore the original group configuration.

Named groups are not maintained when you use a drawing as an external reference or insert it as a block. However, you can bind and then explode the external reference or explode the block to make the group available as an unnamed group.

NOTE Avoid creating large groups containing hundreds or thousands of objects. A large group significantly degrades the performance of this program.

To create a group

1 At the Command prompt, enter **group**.

2 In the Object Grouping dialog box, under Group Identification, enter a group name and a description.

3 In the Create Group area, click New.

   The dialog box closes temporarily.

4 Select objects and press Enter.

5 Click OK.

Command entry: GROUP

Quick Reference

Commands

GROUP

Creates and manages saved sets of objects called groups.

System Variables

PICKSTYLE

Controls the use of group selection and associative hatch selection.

Select Objects in Groups

There are several methods for choosing a group, including selecting the group by name or selecting one of the members of the group.

You can select groups by name at the Select Objects prompt. If the PICKSTYLE system variable is set to 1 or 3 and you select any member of a selectable group, all group members that meet the selection criteria are selected. You can also turn group selection on and off by pressing Ctrl+ H or Shift+Ctrl+A.

All members of selectable groups are also selected when you use object selection cycling (for example, if you want to select an object that lies directly behind another object). Selecting an object that is a member of more than one selectable group selects all the members of all the groups that contain that object. To select groups for editing with grips, use the pointing device to select the group at the Command prompt.

To control whether all grouped objects can be selected individually

Do one of the following:

- At the Command prompt, enter **pickstyle**. Enter 1 to turn on group selection. Objects within groups can be selected as a group only, not individually.
- At the Command prompt, enter **pickstyle**. Enter 0 to turn off group selection. Objects within groups can be selected individually only, not as a group.
- At any time, turn group selection on and off by pressing either Ctrl+H or Ctrl+Shift+A.

To control selectability for a specific group

1. At the Command prompt, enter **group**.
2. In the Object Grouping dialog box, under Group Name, click the group for which you want to change selectability.
3. Under Change Group, click Selectable.

   This changes whether objects within the group are either selectable as a group, subject to the PICKSTYLE system variable. If selectability is turned off, the objects in a group are selectable only as individual objects. At the top of the Object Grouping dialog box, under Selectable, each group displays its current status.
4. Click OK.

Quick Reference

Commands

GROUP

Creates and manages saved sets of objects called groups.

System Variables

PICKSTYLE

Controls the use of group selection and associative hatch selection.

Edit Groups

You can modify groups in a number of ways, including changing their membership, modifying their properties, revising the names and descriptions of groups, and removing them from the drawing.

Edit Objects as a Group

When group selection is turned on, you can move, copy, rotate, and modify groups just as you can modify individual objects. If you need to edit objects within a group, turn off group selection or use grips to edit individual objects. For more information, see Select Objects in Groups on page 1078.

In some circumstances, it is useful to control the order in which objects that belong to the same group are selected. For example, a custom routine that generates toolpaths for numerical control devices might depend on a series of contiguous objects in a specified order.

You can reorder group members in two ways: either change the numerical position of individual members or ranges of group members, or reverse the order of all members. The first object in each group is number 0, not number 1.

Change Group Components, Name, or Description

With the Object Grouping dialog box, you can specify objects to be added to or removed from a group at any time. You can also revise a group's name or description. If deleting an object or removing it from a group leaves the group empty, the group remains defined but without any members.

NOTE Exploding an object such as a block instance or hatch that belongs to a group does not automatically add the resulting components to any group.

Remove Groups

You can delete a group definition by using the “explode” option in the Object Grouping dialog box. This operation is not the same as exploding a block, hatch, or dimension. Objects that belonged to the exploded group remain in the drawing.

As a result, the group is disbanded but the members are not changed in any other way.

To delete a named group

1 At the Command prompt, enter **group**.

2 In the Object Grouping dialog box, select the group name from the list of groups.

3 Under Change Group, select Explode.

4 Click OK.

The group is deleted.

Command entry: GROUP

To reorder group members

1 At the Command prompt, enter **group**.

2 In the Object Grouping dialog box, under Change Group, click Re-Order.

3 In the Order Group dialog box, under Group Name, select the group to reorder.

4 To view the current order of this group, click Highlight.

5 In the Object Grouping dialog box, click Next or Previous to view the objects. Click OK when you have finished viewing the order of the objects.

6 In the Order Group dialog box, under Remove From Position, enter an object number.

7 Under Enter New Position Number for the Object, enter a new position.

8 Under Number of Objects, enter the object number or range of numbers to reorder. Click Re-Order.

9 Click OK to close each dialog box.

Command entry: GROUP

Quick Reference

Commands

GROUP

Creates and manages saved sets of objects called groups.

System Variables

PICKSTYLE

Controls the use of group selection and associative hatch selection.

Correct Mistakes

You can backtrack your recent actions using one of several methods.

Undo a Single Action

The simplest method of backtracking is to use Undo on the Standard toolbar or the U command to undo a single action. Many commands include their own U (undo) option so that you can correct mistakes without leaving the command. When you are creating lines and polylines, for example, enter **u** to undo the last segment.

NOTE By default, the UNDO command is set to combine consecutive pan and zoom commands into a single operation when you undo or redo. However, pan and zoom commands that are started from the menu are not combined, and always remain separate actions.

Undo Several Actions at Once

Use the Mark option of UNDO to mark an action as you work. You can then use the Back option of UNDO to undo all actions that occurred after the marked action. Use the Begin and End options of UNDO to define a set of actions to be treated as a group.

You can also undo several actions at once with the Undo list on the Standard toolbar.

Reverse the Effect of Undo

You can reverse the effect of a single U or UNDO command by using REDO immediately after using U or UNDO.

You can also redo several actions at once with the Redo list on the Standard toolbar.

Erase Objects

You can erase any object that you draw. If you accidentally erase the wrong object, you can use the UNDO command or the OOPS command to restore it.

For more information, see Erase Objects on page 1085.

Cancel a Command

You can cancel a command without completing it by pressing Esc.

To undo the most recent action

■ Click Edit menu ➤ Undo.

Toolbar: Standard
Command entry: U

To undo a specific number of actions

1 On the Standard toolbar, click the Undo list arrow.

A list of actions that you can undo, starting with the most recent action, is displayed.

2 Drag to select the actions to undo.

3 Click to undo the selected actions.

Toolbar: Standard
Command entry: UNDO

To redo an action

■ Click Edit menu ➤ Redo.

Only the action immediately preceding an UNDO command can be reversed with REDO. You cannot use REDO to repeat another command.

Toolbar: Standard
Command entry: REDO

To redo a specific number of actions

1 On the Standard toolbar, click the Redo list arrow.

A list of undo actions that you can redo, starting with the most recent action, is displayed.

2 Drag to select the actions to redo.

3 Click to redo the selected actions.

Toolbar: Standard 
Command entry: MREDO

Quick Reference

Commands

ERASE

Removes objects from a drawing.

OPTIONS

Customizes the program settings.

OOPS

Restores erased objects.

REDO

Reverses the effects of previous UNDO or U command.

MREDO

Reverses the effects of several previous UNDO or U commands.

U

Reverses the most recent operation.

UNDO

Reverses the effect of commands.

System Variables

UNDOCTL

Indicates the state of the Auto, Control, and Group options of the UNDO command.

UNDOMARKS

Stores the number of marks placed in the UNDO control stream by the Mark option.

Erase Objects

There are many ways to delete objects from your drawing and clean up the display.

Remove Unused Definitions, Styles, and Objects

You can remove unused *named and unnamed objects* with PURGE. Some of the unnamed objects you can purge include block definitions, dimension styles, layers, linetypes, and text styles. With PURGE you can also remove zero-length geometry and empty text objects.

Clean Up the Display

You can remove the plus-shaped markers called blips and stray pixels that may be left over from some editing operations from the display area.

- To remove blips, use REDRAW.
- To remove stray pixels, use REGEN.

See also:

- Correct Mistakes on page 1082

To erase an object

1. Click Home tab ➤ Modify panel ➤ Erase.
2. At the Select Objects prompt, use a selection method to select the objects to be erased or enter an option:
    - Enter **L** (Last) to erase the last object drawn.
    - Enter **p** (Previous) to erase the last selection set.
    - Enter **all** to erase all objects from the drawing.
    - Enter **?** to see a list of all selection methods.
3. Press Enter to end the command.

Toolbar: Modify 
Command entry: ERASE

To restore the last erased object

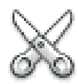
- At the Command prompt, enter **oops**.

The last objects that were removed by ERASE, BLOCK, or WBLOCK are restored.

Command entry: OOPS

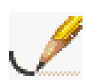
To cut objects to the Clipboard

1 Select the objects you want to cut.

2 Click Home tab ➤ Utilities panel ➤ Cut.  You can also press Ctrl+X.
The objects are available to be pasted into other Windows applications.

Command entry: CUTCLIP

To remove plus-shaped marker blips

- Click View menu ➤ Redraw. 

Command entry: REDRAW

To purge an unused linetype

1 Click Tools tab ➤ Drawing Utilities panel ➤ Purge. 
The Purge dialog box displays a tree view of object types with items that can be purged.

2 To purge unreferenced linetypes, use one of the following methods:
   - To purge all unreferenced linetypes, select Linetypes.
   - To purge specific linetypes, double-click Linetypes to expand the tree view. Then select the linetypes to be purged.

If the item you want to purge is not listed, select View Items You Cannot Purge.

3 You are prompted to confirm each item in the list. If you do not want to confirm each purge, clear the Confirm Each Item to Be Purged option.

4 Click Purge.

To confirm the purging of each item, respond to the prompt by choosing Yes or No, or Yes to All if more than one item is selected.

5 Click Close.

Command entry: PURGE

To remove zero-length geometry and empty text objects

1 Click Tools tab ➤ Drawing Utilities panel ➤ Purge. 

The Purge dialog box displays.

2 Select Purge zero-length geometry and empty text objects.

3 Click Purge.

4 Click Close.

Command entry: PURGE

Quick Reference

Commands

CUTCLIP

Copies selected objects to the Clipboard and removes them from the drawing.

ERASE

Removes objects from a drawing.

OOPS

Restores erased objects.

PURGE

Removes unused items, such as block definitions and layers, from the drawing.

REDRAW

Refreshes the display in the current viewport.

REDRAWALL

Refreshes the display in all viewports.

REGEN

Regenerates the entire drawing from the current viewport.

UNDO

Reverses the effect of commands.

Use Windows Cut, Copy, and Paste

When you want to use objects from a drawing file in another application, you can cut or copy these objects to the Clipboard and then paste them from the Clipboard into the other application.

Cut Objects

Cutting deletes selected objects from the drawing and stores them on the Clipboard. The objects are now available to be pasted into other Microsoft® Windows® documents.

Copy Objects

You can use the Clipboard to copy part or all of a drawing into a document created by another application. The objects are copied in vector format, which retains the high resolution in other applications. These objects are stored in WMF (Windows metafile) format in the Clipboard. The information stored in the Clipboard can then be embedded in the other document. Updating the original drawing does not update the copy embedded in the other application.

Paste Objects

Applications use different internal formats to store Clipboard information. When you copy objects to the Clipboard, information is stored in all available formats. When you paste the Clipboard contents into a drawing, the format that retains the most information is used. However, you can override this setting and convert pasted information to AutoCAD format.

Because it is the easiest format to edit, the AutoCAD format is the preferred format for copying objects to and from AutoCAD. It retains all relevant object information, including block references and 3D aspects.

The Windows metafile (picture) format contains screen vector information, and files can be scaled and printed without losing resolution. Use this format to paste objects into Windows applications that support WMF files. Metafiles pasted into AutoCAD are of higher resolution than bitmapped images (BMP files) but are not as easily manipulated as AutoCAD objects. Bitmapped images are raster images consisting of a pattern of pixels and are commonly used by paint applications.

The color of the object doesn't change when copied to the Clipboard. For example, white objects pasted onto a white background won't be visible. Use the WMFBKGND and WMFFOREGND system variables to control whether the background or foreground is transparent for metafile objects pasted into other applications.

You can insert a linked or embedded object from the Clipboard into a drawing with PASTESPEC. If you convert pasted information to AutoCAD format, the object is inserted as a block reference. To edit the pasted information, explode the block reference into its component objects. When you convert a Windows metafile stored on the Clipboard to AutoCAD format, you may lose some scaling precision. To retain proper scaling, save objects in the original drawing as a block (WBLOCK), and then insert them into AutoCAD using INSERT.

To cut objects to the Clipboard

1. Select the objects you want to cut.
2. Click Home tab ➤ Utilities panel ➤ Cut.

   You can also press Ctrl+X.

   The objects can be pasted into other Windows applications as well.

Command entry: CUTCLIP

To copy objects to the Clipboard

1. Select the objects you want to copy.
2. Click Home tab ➤ Utilities panel ➤ Copy Clip.

You can also press Ctrl+C.

Command entry: COPYCLIP

To paste objects from the Clipboard

- Click Home tab ➤ Utilities panel ➤ Paste. 
  You can also press Ctrl+V.

  The objects currently on the Clipboard are pasted into the drawing.

Command entry: PASTECLIP

To convert pasted information to drawing file format

1. Click Home tab ➤ Utilities panel ➤ Paste Special. 
2. In the Paste Special dialog box, select Paste.
3. From the list of formats, select Picture.
4. Click OK.

Command entry: PASTESPEC

Quick Reference

Commands

COPYBASE

Copies selected objects to the Clipboard along with a specified base point.

COPYCLIP

Copies selected objects to the Clipboard.

CUTCLIP

Copies selected objects to the Clipboard and removes them from the drawing.

PASTEBLOCK

Pastes objects from the Clipboard into the current drawing as a block.

PASTECLIP

Pastes objects from the Clipboard into the current drawing.

PASTEORIG

Pastes objects from the Clipboard into the current drawing using the original coordinates.

PASTESPEC

Pastes objects from the Clipboard into the current drawing and controls the format of the data.

WMFIN

Imports a Windows metafile.

WMFOPTS

Sets options for WMFIN.

WMFOUT

Saves objects to a Windows metafile.

System Variables

OLEHIDE

Controls the display and plotting of OLE objects.

WMFBKGND

Controls the background display when objects are inserted in Windows metafile (WMF) format.

WMFFOREGND

Controls the assignment of the foreground color when objects are inserted in Windows metafile (WMF) format.

Modify Objects

You can easily modify the size, shape, and location of objects.

Choose a Method to Modify Objects

You can easily modify the size, shape, and location of objects. You can

- Enter a command first, and then select the objects to modify.
- Select the objects first, and then enter a command to modify them.

- Select and right-click an object to display a shortcut menu with relevant options.
- Double-click an object to display the Properties palette or, in some cases, a dialog box or editor that is specific to that type of object. (You can specify the double click action for each object type by customizing a CUIx file and loading it into the program.)

See also:

- Change Text on page 1561
- Select Objects on page 1057
- Modify Existing Dimensions on page 1704
- Display and Change the Properties of Objects on page 524

- Work with Custom and Proxy Objects
- "Double Click Actions" in the *Customization Guide*

Quick Reference

Commands

CUI

Manages the customized user interface elements in the product.

DRAGMODE

Controls the way dragged objects are displayed.

PROPERTIES

Controls properties of existing objects.

SELECT

Places selected objects in the Previous selection set.

System Variables

DBLCLKEDIT

Controls the double click editing behavior in the drawing area.

DRAGMODE

Controls the way dragged objects are displayed.

PICKADD

Controls whether subsequent selections replace the current selection set or add to it.

PICKFIRST

Controls whether you select objects before (noun-verb selection) or after you issue a command.

Move or Rotate Objects

You can move objects to a different location, or change the orientation of objects by rotating them by an angle or to other objects.

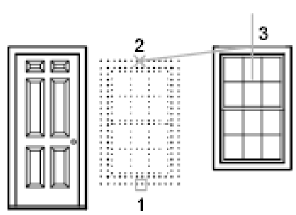
Move Objects

You can move objects at a specified distance and direction from the originals.

Use coordinates, grid snap, object snaps, and other tools to move objects with precision.

Specify Distance with Two Points

Move an object using the distance and direction specified by a base point followed by a second point. In this example, you move the block representing a window. Click Home tab ➤ Modify panel ➤ Move. Then select the object to be moved (1). Specify the base point for the move (2) followed by a second point (3). The object is moved the distance and direction of point 2 to point 3.



Specify Distance with Relative Coordinates

You can move an object using a relative distance by entering coordinate values for the first point and pressing Enter for the second point. The coordinate values are used as a relative displacement rather than the location of a base point.

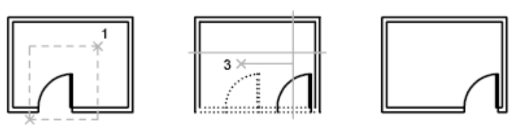
NOTE Do not include an @ sign as you normally would for relative coordinates, because relative coordinates are expected.

To copy objects a specified distance, you can also use direct distance entry with Ortho mode and polar tracking. For more information, see Enter Direct Distances on page 709

Use a Stretch-Move

You can also use STRETCH to move objects if all their endpoints lie entirely within the selection window. Turn on Ortho mode or polar tracking to move the objects at a specific angle.

A practical example is moving a door in a wall. The door in the illustration is entirely within a crossing selection, while the wall lines are only partly within the crossing selection area.



objects selected with crossing selection | door moved with Ortho mode on and direct distance entry | result

The result is that only the endpoints that lie within the crossing selection move.

Use Alternate Methods

You can use grips to move and copy objects quickly. See Use Grips to Edit Objects on page 1141.

You can also select objects and drag them to a new location; press Ctrl to make a copy. Using this method, you can drag objects between open drawings and other applications. If you drag with the right mouse button instead of the left, a shortcut menu is displayed. The menu options include Move Here, Copy Here, Paste as Block, and Cancel. See Embed OLE Objects in Drawings.

To move an object using two points

1 Click Home tab ➤ Modify panel ➤ Move.

2 Select the objects to move.

3 Specify a base point for the move.

4 Specify a second point.

The objects you selected are moved to a new location determined by the distance and direction between the first and second points.

Toolbar: Modify
Command entry: MOVE

To move an object using a displacement

1 Click Home tab ➤ Modify panel ➤ Move.

2 Select the object to move.

3 Enter the displacement in the form of a Cartesian, polar, cylindrical, or spherical coordinate value. Do not include the @ sign, because a relative coordinate is assumed.

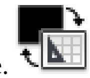
4 At the prompt for the second point, press Enter.

The coordinate values are used as a relative displacement rather than the location of a base point. The selected objects are moved to a new location determined by the relative coordinate values you enter.

Toolbar: Modify
Command entry: MOVE

To move an object from model space to paper space (or vice versa)

1 Click a layout tab.

2 Click Home tab ➤ Modify panel ➤ Change Space. 

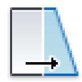
3 Select one or more objects to move.

4 Press Enter.

The object is moved to the new space, and is scaled appropriately to the new space.

Command entry: CHSPACE

To move by stretching

1 Click Home tab ➤ Modify panel ➤ Stretch. 

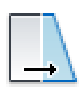
2 Select the object by using crossing selection.

The crossing selection must include at least one vertex or endpoint. Specify crossing selection by clicking, moving your pointing device from right to left, and clicking again.

3 Do *one* of the following:

- Specify the base point for the move, and then specify a second point.
- Enter the displacement in the form of a Cartesian, polar, cylindrical, or spherical coordinate value. Do not include the @ sign, because a relative coordinate is assumed. At the prompt for the second point of displacement, press Enter.

Any objects with at least one vertex or endpoint included within the crossing selection are stretched. Any objects that are completely within the crossing selection are moved without being stretched.

Toolbar: Modify 
Command entry: STRETCH

Quick Reference

Commands

CHSPACE

Moves objects between model space and paper space.

MOVE

Moves objects a specified distance in a specified direction.

PROPERTIES

Controls properties of existing objects.

STRETCH

Stretches objects crossed by a selection window or polygon.

Command Modifiers

Direct Distance Entry (Command Modifier)

Locates the next point at a specified distance in the direction of your cursor.

Rotate Objects

You can rotate objects in your drawing around a specified base point.

To determine the angle of rotation, you can enter an angle value, drag using the cursor, or specify a reference angle to align to an absolute angle.

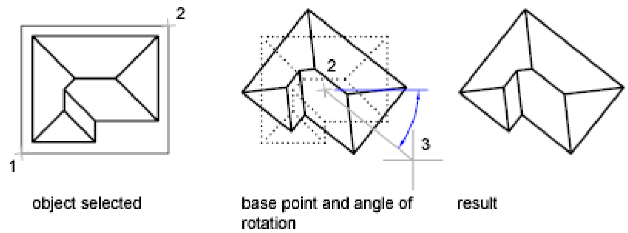
Rotate an Object by a Specified Angle

Enter a rotation angle value from 0 to 360 degrees. You can also enter values in radians, grads, or surveyor bearings. Entering a positive angle value rotates the objects counterclockwise or clockwise, depending on the Direction Control setting in the Drawing Units dialog box.

Rotate an Object by Dragging

Drag the object around the base point and specify a second point. Use Ortho mode, polar tracking, or object snaps for greater precision.

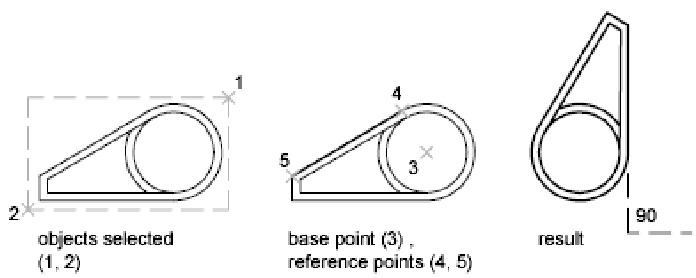
For example, you can rotate the plan view of a house by selecting the objects (1), specifying a base point (2), and specifying an angle of rotation by dragging to another point (3).



Rotate an Object to an Absolute Angle

With the Reference option, you can rotate an object to align it to an absolute angle.

For example, to rotate the part in the illustration so the diagonal edge rotates to 90 degrees, you select the objects to be rotated (1, 2), specify the base point (3), and enter the Reference option. For the reference angle, specify the two endpoints of the diagonal line (4, 5). For the new angle, enter 90.



Rotate an Object in 3D

To rotate 3D objects, you can use either ROTATE or ROTATE3D.

- With ROTATE, you can rotate objects around a specified base point. The axis of rotation passes through the base point and is parallel to the *Z* axis of the current UCS.
- With ROTATE3D, you can specify the axis of rotation using either two points; an object; the *X*, *Y*, or *Z* axis; or the *Z* direction of the current view.

See also:

- Rotate Views in Layout Viewports on page 471

To rotate an object

1. Click Home tab ➤ Modify panel ➤ Rotate.
2. Select the object to rotate.
3. Specify the base point for the rotation.
4. Do *one* of the following:
   - Enter the angle of rotation.
   - Drag the object around its base point and specify a point location to which you want to rotate the object.
   - Enter **c** to create a copy of the selected objects.
   - Enter **r** to rotate the selected objects from a specified reference angle to an absolute angle.

Toolbar: Modify
Command entry: ROTATE

To rotate an object to an absolute angle

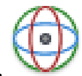
1. Click Home tab ➤ Modify panel ➤ Rotate.
2. Select the objects to rotate.
3. Specify the base point for the rotation.
4. Enter **r** (Reference).
5. Enter a reference angle value or specify two point locations.

   This determines an imaginary line that will be rotated to a new angle.
6. Enter the new angle, or specify a point.

   The value that you enter for the new angle is an absolute angle, not a relative value. Alternatively, if you specify a point, the reference angle will be rotated to that point.

Toolbar: Modify
Command entry: ROTATE

To rotate a 3D object around an axis

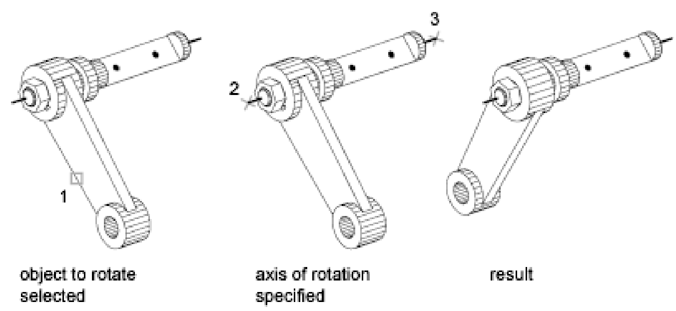
1 Click Modify menu ➤ 3D Operations ➤ Rotate 3D. 

2 Select the object to rotate (1).

3 Specify the start point and endpoint of the axis about which the objects are to be rotated (2 and 3).

   The positive axis direction is from the start point to the end point, and the rotation follows the right-hand rule (see Understand the User Coordinate System in 3D on page 650).

4 Specify the angle of rotation.



Command entry: ROTATE3D

Quick Reference

Commands

ROTATE

Rotates objects around a base point.

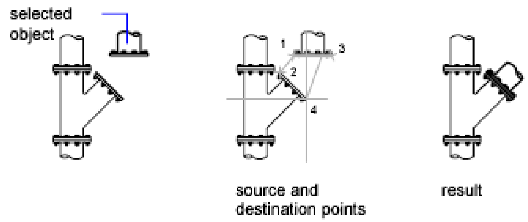
ROTATE3D

Moves objects about a 3D axis.

Align Objects

You can move, rotate, or tilt an object so that it aligns with another object.

In the following example, two pairs of points are used to align the piping in 2D using the ALIGN command. Endpoint object snaps align the pipes precisely.

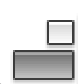


In 3D, use the 3DALIGN command to specify up to three points to define the source plane followed by up to three points to define the destination plane.

- The first source point on an object, called the *base point*, is always moved to the first destination point.
- Specifying a second point for either the source or the destination results in the selected objects being rotated.
- A third point for either the source or the destination results in further rotation of the selected objects.

TIP With 3D solid models, it is recommended that you turn on dynamic UCS to speed the selection of the destination plane.

To align two objects in 2D

1 Click Modify menu ➤ 3D Operations ➤ Align. 

2 Select the objects that you want to align.

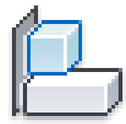
3 Specify a source point and then the corresponding destination point. To rotate the object, specify a second source point followed by a second destination point.

4 Press Enter to end the command.

The selected objects are moved from the source point to the destination point, and second and third points, if you specify them, rotate, and tilt the selected objects.

Command entry: ALIGN

To align two objects in 3D

1. Click Home tab ➤ Modify panel ➤ 3D Align. 
2. Select the objects that you want to align.
3. Specify either one, two, or three source points and then the corresponding first, second, or third destination points. The first point is called the *base point*.

   The selected objects are moved from the source point to the destination point, and second and third points, if you specify them, rotate, and tilt the selected objects.

Command entry: 3DALIGN

Quick Reference

Commands

3DALIGN

Aligns objects with other objects in 2D and 3D.

ALIGN

Aligns objects with other objects in 2D and 3D.

System Variables

UCSDETECT

Controls whether dynamic UCS acquisition is active or not.

Copy, Offset, or Mirror Objects

You can create duplicates of objects in your drawing that are either identical or similar to selected objects.

Copy Objects

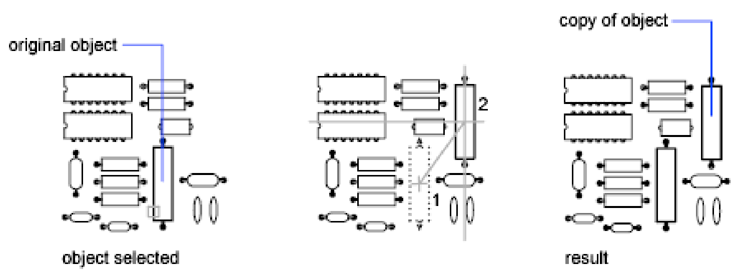
You can create duplicates of objects at a specified distance and direction from the originals.

Use coordinates, grid snap, object snaps, and other tools to copy objects with precision.

You can also use grips to move and copy objects quickly. See Use Grips to Edit Objects on page 1141.

Specify Distance with Two Points

Copy an object using the distance and direction specified by a base point followed by a second point. In this example, you copy the block representing an electronic component. Click Edit menu ➤ Copy.Then select the original object to be copied. Specify the base point for the move (1) followed by a second point (2). The object is copied the distance and direction of point 1 to point 2.



Specify Distance with Relative Coordinates

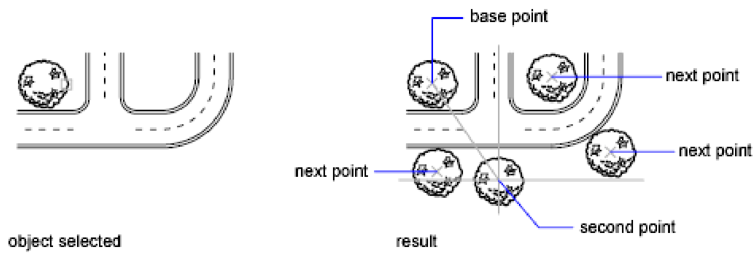
Copy an object using a relative distance by entering coordinate values for the first point and pressing Enter for the second point. The coordinate values are used as a relative displacement rather than the location of a base point.

NOTE Do not include an @ sign as you normally would for relative coordinates, because relative coordinates are expected.

To copy objects a specified distance, you can also use direct distance entry with Ortho mode and polar tracking. For more information, see Enter Direct Distances on page 709.

Create Multiple Copies

The COPY command repeats automatically by default. To exit the command, press Enter. To change the default, use the COPYMODE system variable.



Move and Copy Objects by Dragging

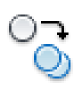
You can also select objects and drag them to a new location using the left mouse button over one of the selected objects; press Ctrl to make a copy. Using this method, you can drag objects between open drawings and other applications.

If you drag with the right mouse button instead of the left, a shortcut menu is displayed after you drag the objects. The menu options include Move Here, Copy Here, Paste as Block, and Cancel.

For information about using object linking and embedding, see Embed OLE Objects in Drawings.

To copy an object using two points

1. Click Home tab ➤ Modify panel ➤ Copy.
2. Select the objects to copy.
3. Specify the base point.
4. Specify the second point. Press Enter.

Toolbar: Modify 

Command entry: COPY

Quick Reference

Commands

COPY

Copies objects a specified distance in a specified direction.

System Variables

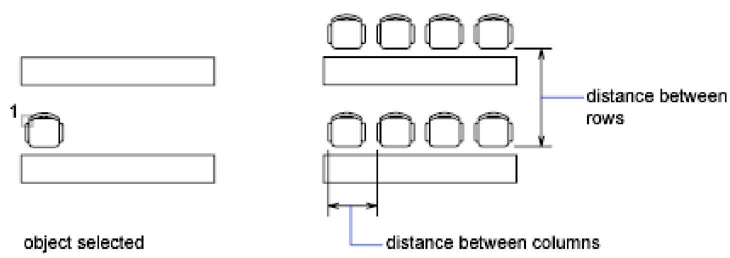
COPYMODE

Controls whether the COPY command repeats automatically.

Create an Array of Objects

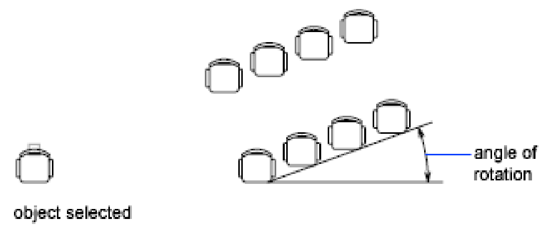
You can create copies of objects in a rectangular or polar (circular) pattern called an array.

For rectangular arrays, you control the number of rows and columns and the distance between each. For polar arrays, you control the number of copies of the object and whether the copies are rotated. To create many regularly spaced objects, arraying is faster than copying.



Create Rectangular Arrays

A rectangular array is built along a baseline defined by the current snap rotation angle. This angle is zero by default, so the rows and columns of a rectangular array are orthogonal with respect to the *X* and *Y* axes. The default angle 0 direction setting can be changed in UNITS.



Create Polar Arrays

When you create a polar array, the array is drawn counterclockwise or clockwise, depending on whether you enter a positive or a negative value for the angle to fill.

The radius of the array is determined by the distance from the specified center point to a *reference* or base point on the last selected object. You can use the default reference point (usually an arbitrary point that coincides with a snap point), or you can specify a new base point to be used as the reference point.

Array in 3D

With 3DARRAY, you can create a rectangular array or a polar array of objects in 3D. In addition to specifying the number of columns (*X* direction) and rows (*Y* direction), you also specify the number of levels (*Z* direction).

Limit the Size of Arrays

If you specify a very large number of rows and columns for an array, it may take a long time to create the copies. By default, the number of array elements that can be generated by one command is limited to approximately 100,000. This limit is controlled by the MaxArray setting in the registry.

You can change the limit by setting the MaxArray system registry variable using (**setenv "MaxArray" "n"**) where *n* is a number between 100 and 10000000 (ten million).

NOTE When changing the value of MaxArray, you must enter MaxArray with the capitalization shown.

To create a rectangular array

1 Click Home tab ➤ Modify panel ➤ Array.

2 In the Array dialog box, select Rectangular Array.

3 Click Select Objects.

The Array dialog box closes. You are prompted for object selection.

4 Select the objects to be arrayed and press Enter.

5 In the Rows and Columns boxes, enter the number of rows and columns in the array.

6 Specify the horizontal and vertical spacing (offsets) between objects by using one of the following methods:

- In the Row Offset and Column Offset boxes, enter the distance between rows and between columns. Adding a plus sign (+) or a minus sign (-) determines direction.
- Click the Pick Both Offsets button to use the pointing device to specify the diagonal corners of a cell in the array. The cell determines the vertical and horizontal spacing of the rows and columns.
- Click the Pick Row Offset or Pick Column Offset button to use the pointing device to specify the horizontal and vertical spacing.

The example box displays the result.

7 To change the rotation angle of the array, enter the new angle next to Angle of Array.

8 The default angle 0 direction setting can also be changed in UNITS.

9 Click OK to create the array.

Toolbar: Modify
Command entry: ARRAY

To create a polar array

1 Click Home tab ➤ Modify panel ➤ Array.

2 In the Array dialog box, select Polar Array.

3 Next to Center Point, do one of the following:
- Enter an *X* value and a *Y* value for the center point of the polar array.
- Click the Pick Center Point button. The Array dialog box closes and you are prompted for object selection. Use the pointing device to specify the center point of the polar array.

4 Click Select Objects.

The Array dialog box closes and you are prompted for object selection.

5 Select the objects to be arrayed.

6 In the Method box, select one of the following methods:
- Total Number of Items & Angle to Fill
- Total Number of Items & Angle Between Items
- Angle to Fill & Angle Between Items

7 Enter the number of items (including the original object), if available.

8 Use one of the following methods:
- Enter the angle to fill and angle between items, if available. Angle to Fill specifies the distance to fill around the circumference of the array. Angle Between Items specifies the distance between each item.
- Click the Pick Angle to Fill button and the Pick Angle Between Items button. Use the pointing device to specify the angle to fill and the angle between items.

The example box displays the result.

9 You can set any of the following options:
- To rotate the objects as they are arrayed, select Rotate Items As Copied. The example area displays the result.
- To specify the *X,Y* base point, select More, clear the Set to Object's Default option and enter values in the *X* and *Y* boxes, or click the Pick Base Point button and use the pointing device to specify the point.

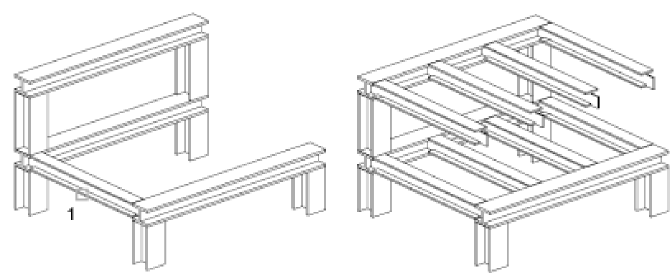
10 Click OK to create the array.

Toolbar: Modify

Command entry: ARRAY

To create a 3D rectangular array of objects

1. Click Home tab ➤ Modify panel ➤ 3D Array.
2. Select the object to array (1).
3. Specify Rectangular.
4. Enter the number of rows.
5. Enter the number of columns.
6. Enter the number of levels.
7. Specify the distance between rows.
8. Specify the distance between columns.
9. Specify the distance between levels.



object to array selected   result

Command entry: 3DARRAY

To create a 3D polar array of objects

1. Click Home tab ➤ Modify panel ➤ 3D Array.
2. Select the object to array (1).

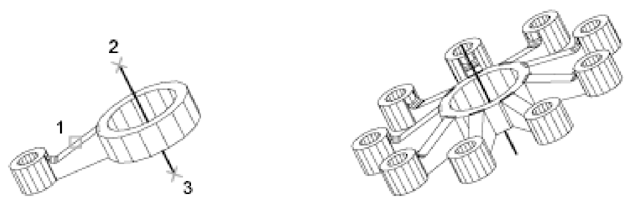
3 Specify Polar.

4 Enter the number of items to array.

5 Specify the angle that the arrayed objects are to fill.

6 Press Enter to rotate the objects as they are arrayed, or enter **n** to retain their orientation.

7 Specify the start point and endpoint of the axis about which the objects are to be rotated (2 and 3).



Command entry: 3DARRAY

Quick Reference

Commands

3DARRAY

Creates a 3D matrix of objects in a rectangular or polar arrangement.

ARRAY

Creates multiple copies of objects in a pattern.

DSETTINGS

Sets grid and snap, polar and object snap tracking, object snap modes, Dynamic Input, and Quick Properties.

UCS

Manages user coordinate systems.

UNITS

Controls coordinate and angle display formats and precision.

System Variables

ANGBASE

Sets the base angle to 0 with respect to the current UCS.

ANGDIR

Sets the direction of positive angles.

SNAPANG

Sets the snap and grid rotation angle for the current viewport relative to the current UCS.

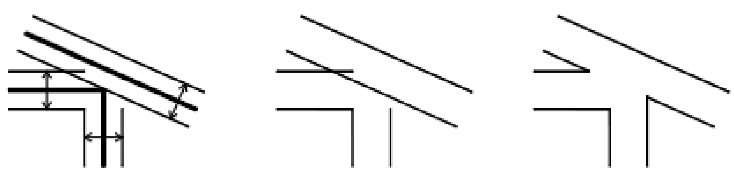
Offset an Object

Offset an object to create a new object whose shape parallels the shape of the original object.

OFFSET creates a new object whose shape parallels the shape of a selected object. Offsetting a circle or an arc creates a larger or smaller circle or arc, depending on which side you specify for the offset.

polyline　　polyline with offset

A highly effective drawing technique is to offset objects and then trim or extend their ends.



offset　　trim and extend the offset lines　　result

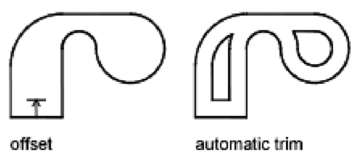
You can offset

- Lines

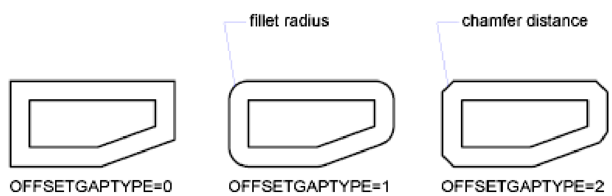
- Arcs
- Circles
- Ellipses and elliptical arcs (resulting in an oval-shaped spline)
- 2D polylines
- Construction lines (xlines) and rays
- Splines

Special Cases for Offset Polylines and Splines

2D polylines and splines are trimmed automatically when the offset distance is larger than can otherwise be accommodated.



Closed 2D polylines that are offset to create larger polylines result in potential gaps between segments. The OFFSETGAPTYPE system variable controls how these potential gaps are closed.



To offset an object by specifying a distance

1. Click Home tab ➤ Modify panel ➤ Offset.
2. Specify the offset distance.
   You can enter a value or use the pointing device.
3. Select the object to offset.
4. Specify a point on the side where you want to place the new objects.
5. Select another object to offset, or press Enter to end the command.

Toolbar: Modify 
Command entry: OFFSET

To offset an object through a point

1 Click Home tab ➤ Modify panel ➤ Offset.

2 Enter **t** (Through).

3 Select the object to offset.

4 Specify the through point.

5 Select another object to offset, or press Enter to end the command.

Toolbar: Modify 
Command entry: OFFSET

Quick Reference

Commands

OFFSET

Creates concentric circles, parallel lines, and parallel curves.

System Variables

OFFSETDIST

Sets the default offset distance.

OFFSETGAPTYPE

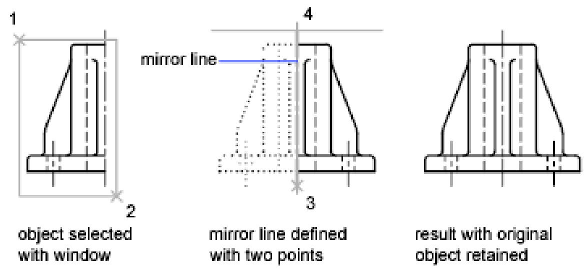
Controls how potential gaps between segments are treated when closed polylines are offset.

Mirror Objects

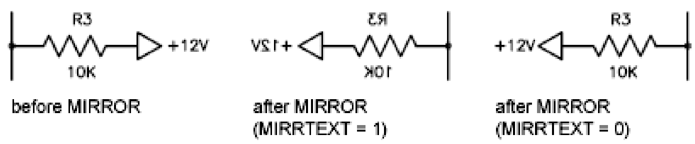
You can flip objects about a specified axis to create a symmetrical mirror image.

Mirroring is useful for creating symmetrical objects because you can quickly draw half the object and then mirror it instead of drawing the entire object.

You flip objects about an axis called a mirror line to create a mirror image. To specify this temporary mirror line, you enter two points. You can choose whether to erase or retain the original objects.



By default, when you mirror text, attributes, and attribute definitions, they are not reversed or turned upside down in the mirror image. The text has the same alignment and justification as before the object was mirrored. If you do want text to be reversed, set the MIRRTEXT system variable to 1.

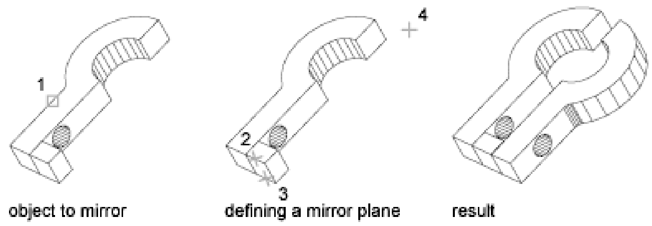


MIRRTEXT affects text that is created with the TEXT, ATTDEF, or MTEXT commands; attribute definitions; and variable attributes. Text and constant attributes that are part of an inserted block are reversed when the block is mirrored regardless of the MIRRTEXT setting.

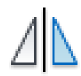
Mirror in 3D

With MIRROR3D, you can mirror objects across a specified mirroring plane. The mirroring plane can be one of the following:

- The plane of a planar object
- A plane parallel to the *XY*, *YZ*, or *XZ* plane of the current UCS that passes through a specified point
- A plane defined by three specified points (2, 3, and 4)

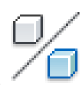


To mirror objects in 2D

1 Click Home tab ➤ Modify panel ➤ Mirror. 
2 Select the objects to mirror.
3 Specify the first point of the mirror line.
4 Specify the second point.
5 Press Enter to retain the original objects, or enter **y** to erase them.

Command entry: MIRROR

To mirror objects in 3D

1 Click Home tab ➤ Modify panel ➤ 3D Mirror. 
2 Select the object to mirror.
3 Specify three points to define a mirroring plane.
4 Press Enter to retain the original objects, or enter **y** to delete them.

Command entry: MIRROR3D

Quick Reference

Commands

MIRROR

Creates a mirrored copy of selected objects.

MIRROR3D

Creates a mirrored copy of selected objects across a mirroring plane.

System Variables

MIRRTEXT

Controls how the MIRROR command reflects text.

Change the Size and Shape of Objects

There are several methods for adjusting the lengths of existing objects relative to other objects, both symmetrically and asymmetrically.

Trim or Extend Objects

You can shorten or lengthen objects to meet the edges of other objects.

This means you can first create an object such as a line and then later adjust it to fit exactly between other objects.

Objects you select as cutting edges or boundary edges are not required to intersect the object being trimmed. You can trim or extend an object to a projected edge or to an extrapolated intersection; that is, where the objects would intersect if they were extended.

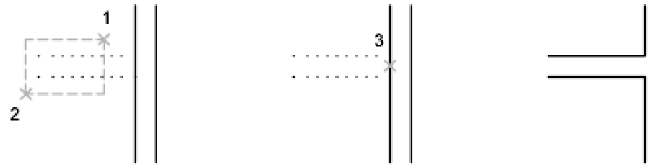
If you do not specify a boundary and press Enter at the Select Objects prompt, all displayed objects become potential boundaries.

NOTE To select cutting edges or boundary edges that include blocks, you can use only the single selection, Crossing, Fence, and Select All options.

Trim Objects

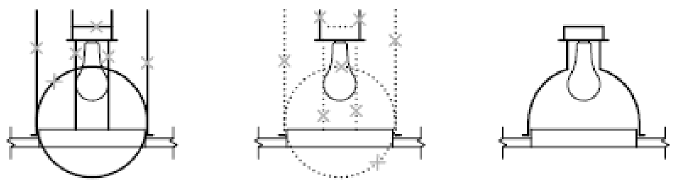
You can trim objects so that they end precisely at boundary edges defined by other objects.

For example, you can clean up the intersection of two walls smoothly by trimming.



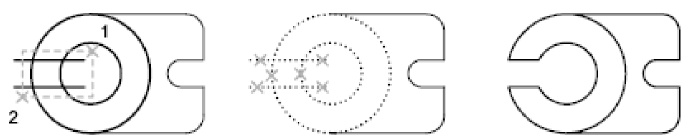
cutting edges selected with a crossing selection | object to trim selected | result

An object can be one of the cutting edges and one of the objects being trimmed. For example, in the illustrated light fixture, the circle is a cutting edge for the construction lines and is also being trimmed.



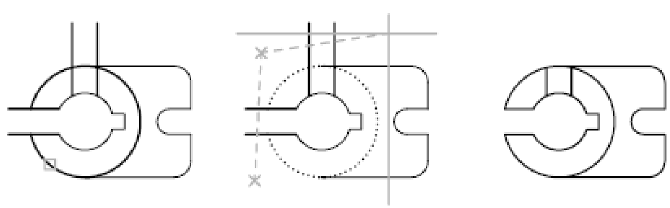
cutting edges selected | objects to trim selected | result

When you trim several objects, the different selection methods can help you choose the current cutting edges and objects to trim. In the following example, the cutting edges are selected using crossing selection.



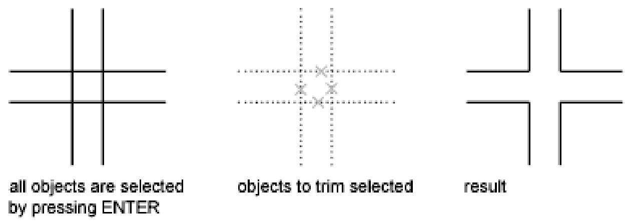
edges selected with crossing selection | objects to trim selected | result

The following example uses the fence selection method to select a series of objects for trimming.



cutting edge selected | objects to trim selected with fence selection | result

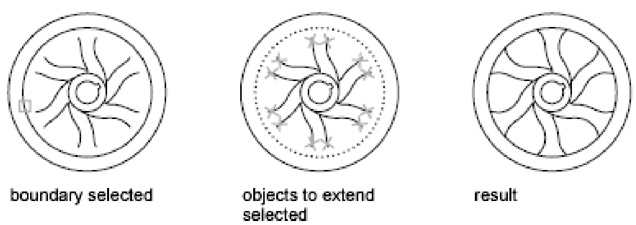
You can trim objects to their nearest intersection with other objects. Instead of selecting cutting edges, you press Enter. Then, when you select the objects to trim, the nearest displayed objects act as cutting edges. In this example, the walls are trimmed so that they intersect smoothly.



You can extend objects without leaving the TRIM command. Hold down Shift and select the objects to be extended.

Extend Objects

Extending operates the same way as trimming. You can extend objects so they end precisely at boundary edges defined by other objects. In this example, you extend the lines precisely to a circle, which is the boundary edge.

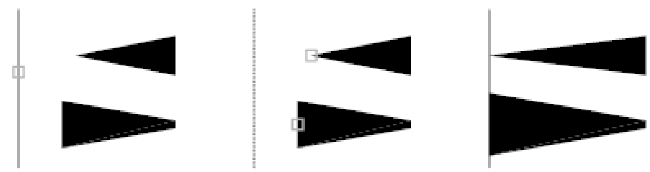


You can trim objects without leaving the EXTEND command. Hold down Shift and select the objects to be trimmed.

Trim and Extend Wide Polylines

2D wide polylines trim and extend at their centerlines. The ends of wide polylines are always square. Trimming a wide polyline at an angle causes portions of the end to extend beyond the cutting edge

If you trim or extend a tapered 2D polyline segment, the width of the extended end is corrected to continue the original taper to the new endpoint. If this correction gives the segment a negative ending width, the ending width is forced to 0.



Trim and Extend Spline-Fit Polylines

Trimming a spline-fit polyline removes the curve-fit information and changes the spline-fit segments into ordinary polyline segments.

Extending a spline-fit polyline adds a new vertex to the control frame for the polyline.

Trim or Extend in 3D

You can trim or extend an object to any other object in 3D space, regardless of whether the objects are on the same plane or parallel to the cutting or boundary edges. In the TRIM and EXTEND commands, use the Project and Edge options to select one of three projections for trimming or extending:

- The *XY* plane of the current UCS
- The plane of the current view
- True 3D, which is not a projection

See also:

- Break and Join Objects on page 1138

To extend an object

1. Click Home tab ➤ Modify panel ➤ Extend.
2. Select the objects to serve as boundary edges.

   To select all displayed objects as potential boundary edges, press Enter without selecting any objects.
3. Select the objects to extend.

Toolbar: Modify

Command entry: EXTEND

To trim an object

1 Click Home tab ➤ Modify panel ➤ Trim.

2 Select the objects to serve as cutting edges.

   To select all displayed objects as potential cutting edges, press Enter without selecting any objects.

3 Select the objects to trim.

Toolbar: Modify

Command entry: TRIM

To extend objects in 3D wireframe models

1 Click Home tab ➤ Modify panel ➤ Extend .

2 Select the boundary edge for extending (1).

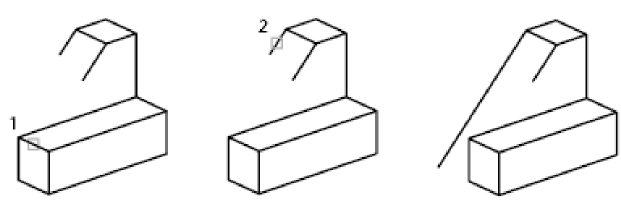
3 Enter **e** (Edge).

4 Enter **e** (Extend).

5 Enter **p** (Project).

6 Enter **u** (UCS).

7 Select the object to extend (2).



Command entry: EXTEND

To trim in 3D using the current view plane

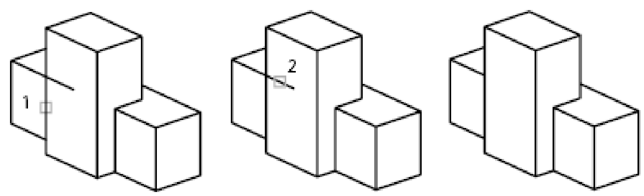
1 Click Home tab ➤ Modify panel ➤ Trim  .

2 Select the cutting edge for trimming (1).

3 Enter **p** (Project).

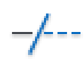
4 Enter **v** (View).

5 Select the object to trim (2).



Command entry: TRIM

To trim objects in 3D wireframe models

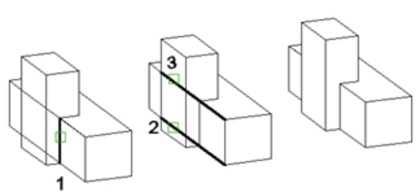
1 Click Home tab ➤ Modify panel ➤ Trim  .

2 Select the cutting edges to use for trimming (1).

3 Enter **p** (Project).

4 Enter **n** (None).

5 Select the object to trim (2 and 3).



Command entry: TRIM

Quick Reference

Commands

BREAK

Breaks the selected object between two points.

EXTEND

Extends objects to meet the edges of other objects.

JOIN

Joins similar objects to form a single, unbroken object.

LENGTHEN

Changes the length of objects and the included angle of arcs.

PROPERTIES

Controls properties of existing objects.

TRIM

Trims objects to meet the edges of other objects.

System Variables

EDGEMODE

Controls how the TRIM and EXTEND commands determine cutting and boundary edges.

PROJMODE

Sets the current Projection mode for trimming or extending.

Resize or Reshape Objects

You can resize objects to make them longer or shorter in only one direction or to make them proportionally larger or smaller.

You can also stretch certain objects by moving an endpoint, vertex, or control point.

Lengthen Objects

With LENGTHEN, you can change the included angle of arcs and the length of the following objects:

- Lines
- Arcs
- Open polylines
- Elliptical arcs
- Open splines.

The results are similar to extending and trimming. You can

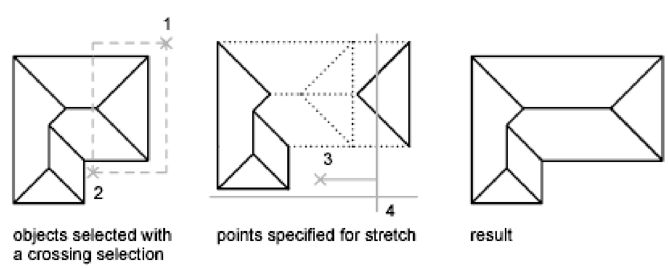
- Drag an object endpoint dynamically
- Specify a new length or angle as a percentage of the total length or angle
- Specify an incremental length or angle measured from an endpoint
- Specify the object's total absolute length or included angle

Stretch Objects

With STRETCH, you relocate the endpoints of objects that lie across or within a crossing selection window.

- Objects that are partially enclosed by a crossing window are stretched.
- Objects that are completely enclosed within the crossing window, or that are selected individually, are moved rather than stretched.

To stretch an object, you specify a base point and then a point of displacement.



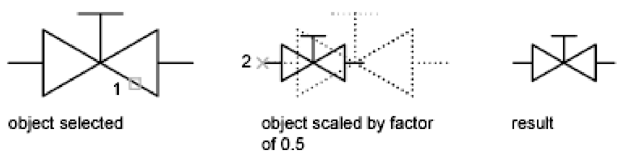
To stretch with precision, use object snaps, grid snaps, and relative coordinate entry.

Scale Objects Using a Scale Factor

With SCALE, you can make an object uniformly larger or smaller. To scale an object, you specify a base point and a scale factor. Alternatively, you can specify a length to be used as a scale factor based on the current drawing units.

A scale factor greater than 1 enlarges the object. A scale factor between 0 and 1 shrinks the object.

Scaling changes the size of all dimensions of the selected object. A scale factor greater than 1 enlarges the object. A scale factor less than 1 shrinks the object.



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.

Scale Objects Using a Reference Distance

You can also scale by reference. Scaling by reference uses an existing distance as a basis for the new size. To scale by reference, specify the current distance and then the new desired size. For example, if one side of an object is 4.8 units long and you want to expand it to 7.5 units, use 4.8 as the reference length.

You can use the Reference option to scale an entire drawing. For example, use this option when the original drawing units need to be changed. Select all objects in the drawing. Then use Reference to select two points and specify the intended distance. All the objects in the drawing are scaled accordingly.

See also:

- Break and Join Objects on page 1138

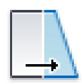
To stretch an object

1. Click Home tab ➤ Modify panel ➤ Stretch.
2. Select the object using a crossing window selection.

The crossing window must include at least one vertex or endpoint.

3 Do *one* of the following:
- Enter the displacement in the form of a relative Cartesian, polar, cylindrical, or spherical coordinate. Do not include the @ sign, because a relative coordinate is assumed. Press Enter at the prompt for the second point of displacement.
- Specify the base point for the stretch, and then specify a second point, to determine the distance and direction.

Any objects with at least one vertex or endpoint included within the crossing window are stretched. Any objects that are completely within the crossing window, or selected individually, are moved without stretching.

Toolbar: Modify 
Command entry: STRETCH

To scale an object by a scale factor

1 Click Home tab ➤ Modify panel ➤ Scale.

2 Select the object to scale.

3 Specify the base point.

4 Enter the scale factor or drag and click to specify a new scale.

Toolbar: Modify 
Command entry: SCALE

To scale an object by reference

1 Click Home tab ➤ Modify panel ➤ Scale.

2 Select the object to scale.

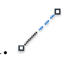
3 Select the base point.

4 Enter **r** (Reference).

5 Select the first and second reference points, or enter a value for the reference length.

Toolbar: Modify 
Command entry: SCALE

To change the length of an object by dragging

1 Click Home tab ➤ Modify panel ➤ Lengthen. 

2 Enter **dy** (Dynamic Dragging mode).

3 Select the object you want to lengthen.

4 Drag the endpoint closest to the point of selection, and specify a new endpoint.

   The selected object is lengthened or shortened without changing its location or orientation.

Command entry: LENGTHEN

Quick Reference

Commands

JOIN

Joins similar objects to form a single, unbroken object.

LENGTHEN

Changes the length of objects and the included angle of arcs.

PEDIT

Edits polylines and 3D polygon meshes.

PROPERTIES

Controls properties of existing objects.

SCALE

Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling.

SPLINEDIT

Edits a spline or spline-fit polyline.

STRETCH

Stretches objects crossed by a selection window or polygon.

System Variables

PLINECONVERTMODE

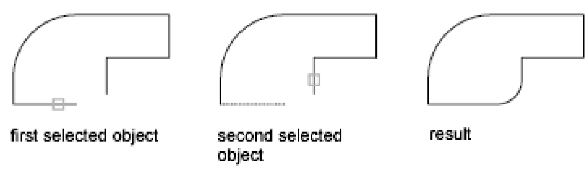
Specifies the fit method used in converting splines to polylines.

Fillet, Chamfer, Break, or Join Objects

You can change objects to meet in rounded or flattened corners. You can also create or close gaps in objects.

Create Fillets

A fillet connects two objects with an arc that is tangent to the objects and has a specified radius.



An inside corner is called a fillet and an outside corner is called a round; you can create both using the FILLET command.

You can fillet

- Arcs
- Circles
- Ellipses and elliptical arcs
- Lines
- Polylines
- Rays

- Splines
- Xlines
- 3D solids

FILLET can be used to round all corners on a polyline using a single command.

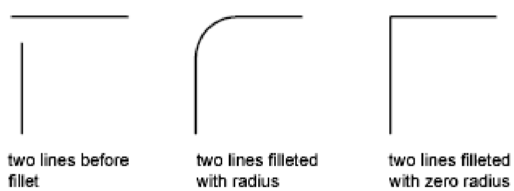
NOTE Filleting a hatch boundary that was defined from line segments removes hatch associativity. If the hatch boundary was defined from a polyline, associativity is maintained.

If both objects being filleted are on the same layer, the fillet arc is created on that layer. Otherwise, the fillet arc is created on the current layer. The layer affects object properties including color and linetype.

Use the Multiple option to fillet more than one set of objects without leaving the command.

Set the Fillet Radius

The fillet radius is the radius of the arc that connects filleted objects. Changing the fillet radius affects subsequent fillets. If you set the fillet radius to 0, filleted objects are trimmed or extended until they intersect, but no arc is created.



You can hold down Shift while selecting the objects to override the current fillet radius with a value of 0.

Trim and Extend Filleted Objects

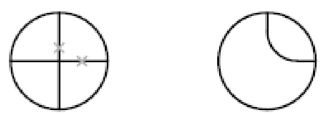
You can use the Trim option to specify whether the selected objects are trimmed or extended to the endpoints of the resulting arc or left unchanged.

two lines filleted with Trim option set

two lines filleted with No Trim option set

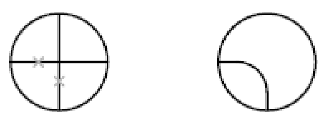
Control the Location of the Fillet

Depending on the locations you specify, more than one possible fillet can exist between the selected objects. Compare the selection locations and resulting fillets in the illustrations.



fillet location points selected

result

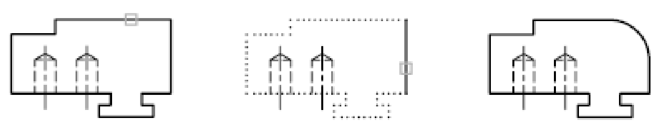


fillet location points selected

result

Fillet Line and Polyline Combinations

To fillet lines with polylines, each line or its extension must intersect one of the polyline line segments. If the Trim option is on, the filleted objects and the fillet arc join to form a single new polyline.



polyline selected

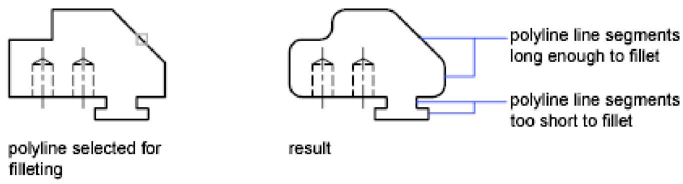
line selected

result

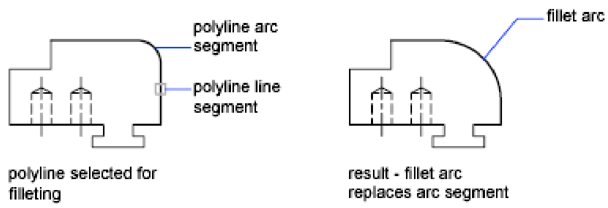
Fillet an Entire Polyline

You can fillet an entire polyline or remove fillets from an entire polyline.

If you set a nonzero fillet radius, FILLET inserts fillet arcs at the vertex of each polyline segment that is long enough to accommodate the fillet radius.



If two polyline line segments converge as they approach an arc segment that separates them, FILLET removes the arc segment and replaces it with a fillet arc.

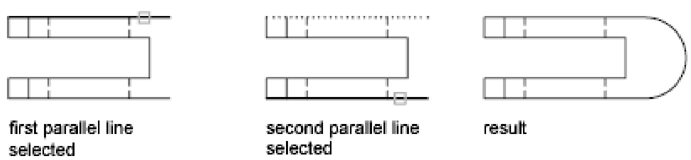


If you set the fillet radius to 0, no fillet arcs are inserted. If two polyline line segments are separated by one arc segment, FILLET removes that arc and extends the lines until they intersect.

Fillet Parallel Lines

You can fillet parallel lines, xlines, and rays. The current fillet radius temporarily adjusts to create an arc that is tangent to both objects and located in the plane common to both objects.

The first selected object must be a line or a ray, but the second object can be a line, an xline, or a ray. The fillet arc connects as shown in the illustration.



Fillet Objects with Non-Zero Thickness in 3D

You can fillet coplanar objects with extrusion directions not parallel to the *Z* axis of the current UCS. FILLET determines the extrusion direction for the fillet arc in 3D space closest to the direction of the *Z* axis of the current UCS.

To set the fillet radius

1. Click Home tab ➤ Modify panel ➤ Fillet.
2. Enter **r** (Radius).
3. Enter the fillet radius
4. Select the objects to fillet.

Toolbar: Modify 
Command entry: FILLET

To fillet two line segments

1. Click Home tab ➤ Modify panel ➤ Fillet.
2. Select the first line.
3. Select the second line.

Toolbar: Modify 
Command entry: FILLET

To fillet without trimming

1. Click Home tab ➤ Modify panel ➤ Fillet.
2. If necessary, enter **t** (Trim). Enter **n** (No Trim).
3. Select the objects to fillet.

Toolbar: Modify 
Command entry: FILLET

To fillet an entire polyline

1. Click Home tab ➤ Modify panel ➤ Fillet.
2. Enter **p** (Polyline).
3. Select the polyline.

Toolbar: Modify 
Command entry: FILLET

To fillet multiple sets of objects

1. Click Home tab ➤ Modify panel ➤ Fillet.
2. Enter **m** (Multiple).

   The main prompt is displayed.
3. Select the first line, or enter an option and complete the prompts for that option. Select the first line.
4. Select the second line.

   The main prompt is displayed again.
5. Select the first line for the next fillet, or press Enter or Esc to end the command.

Toolbar: Modify 
Command entry: FILLET

Quick Reference

Commands

FILLET

Rounds and fillets the edges of objects.

System Variables

FILLETRAD

Stores the current fillet radius.

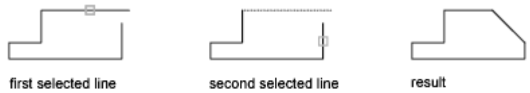
TRIMMODE

Controls whether selected edges for chamfers and fillets are trimmed.

Create Chamfers

A chamfer connects two objects to meet in a flattened or beveled corner.

A chamfer connects two objects with an angled line. It is usually used to represent a beveled edge on a corner.



You can chamfer

- Lines
- Polylines
- Rays
- Xlines
- 3D solids

CHAMFER can be used to bevel all corners of a polyline using a single command.

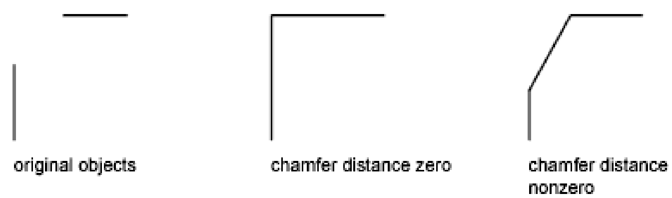
NOTE Chamfering a hatch boundary that was defined from line segments removes hatch associativity. If the hatch boundary was defined from a polyline, associativity is maintained.

If both objects being chamfered are on the same layer, the chamfer line is created on that layer. Otherwise, the chamfer line is created on the current layer. The layer affects object properties including color and linetype.

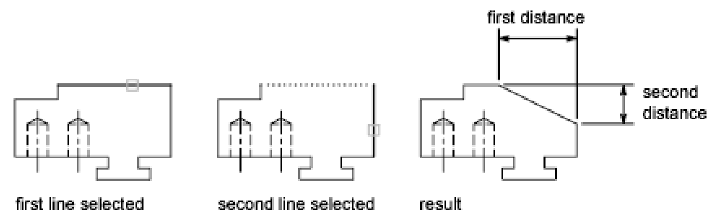
Use the Multiple option to chamfer more than one set of objects without leaving the command.

Chamfer by Specifying Distances

The chamfer distance is the amount each object is trimmed or extended to meet the chamfer line or to intersect the other. If both chamfer distances are 0, chamfering trims or extends the two objects until they intersect but does not create a chamfer line. You can hold down Shift while selecting the objects to override the current chamfer distances with a value of 0.



In the following example, you set the chamfer distance to 0.5 for the first line and 0.25 for the second line. After you specify the chamfer distance, you select the two lines as shown.



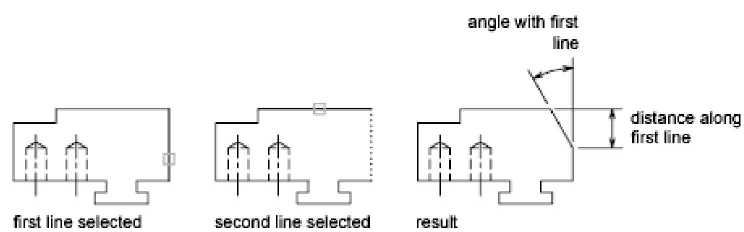
Trim and Extend Chamfered Objects

By default, objects are trimmed when chamfered, but you can use the Trim option to specify that they remain untrimmed.

Chamfer by Specify Length and Angle

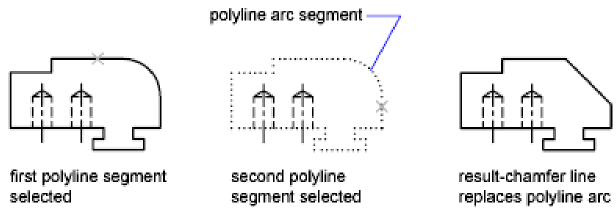
You can chamfer two objects by specifying where on the first selected object the chamfer line starts, and then the angle the chamfer line forms with this object.

In this example, you chamfer two lines so that the chamfer line starts 1.5 units from the intersection along the first line and forms an angle of 30 degrees with this line.



Chamfer Polylines and Polyline Segments

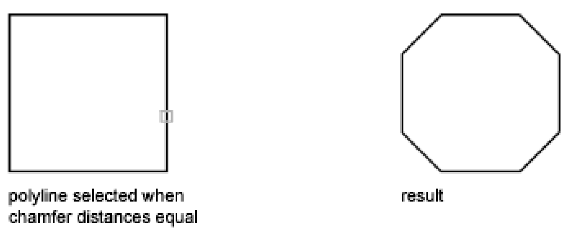
If the two objects you select for chamfering are segments of a polyline, they must be adjacent or separated by no more than one arc segment. If they are separated by an arc segment, as shown in the illustration, chamfering deletes the arc and replaces it with a chamfer line.



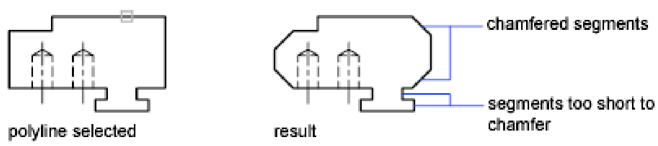
Chamfer an Entire Polyline

When you chamfer an entire polyline, each intersection is chamfered. For best results, keep the first and second chamfer distances equal.

In this example, the chamfer distances are set to equal values.



When you chamfer an entire polyline, only the segments that are long enough to accommodate the chamfer distance are chamfered. The polyline in the following illustration has some segments too short to be chamfered.



To set chamfer distances

1 Click Home tab ➤ Modify panel ➤ Chamfer.

2 Enter **d** (Distances).

3 Enter the first chamfer distance.

4 Enter the second chamfer distance.

5 Select the lines for chamfering.

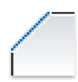
Toolbar: Modify 
Command entry: CHAMFER

To chamfer two nonparallel line segments

1 Click Home tab ➤ Modify panel ➤ Chamfer.

2 Select the first line.

3 Select the second line.

Toolbar: Modify 
Command entry: CHAMFER

To chamfer by specifying chamfer length and angle

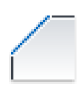
1 Click Home tab ➤ Modify panel ➤ Chamfer.

2 Enter **a** (Angle).

3 Enter the distance from the corner to be chamfered along the first line.

4 Enter the chamfer angle.

5 Select the first line. Then select the second line.

Toolbar: Modify 
Command entry: CHAMFER

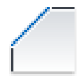
To chamfer without trimming

1 Click Home tab ➤ Modify panel ➤ Chamfer.

2 Enter **t** (Trim Control).

3 Enter **n** (No Trim).

4 Select the objects to chamfer.

Toolbar: Modify 
Command entry: CHAMFER

To chamfer an entire polyline

1 Click Home tab ➤ Modify panel ➤ Chamfer.

2 Enter **p** (Polyline).

3 Select the polyline.

The polyline is chamfered using the current chamfer method and the default distances.

Toolbar: Modify 
Command entry: CHAMFER

To chamfer multiple sets of objects

1 Click Home tab ➤ Modify panel ➤ Chamfer.

2 Enter **m** (Multiple).

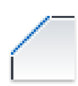
The main prompt is displayed.

3 Select the first line, or enter an option and complete the prompts for that option and then select the first line.

4 Select the second line.

The main prompt is displayed again.

5 Select the first line for the next chamfer, or press Enter or Esc to end the command.

Toolbar: Modify 
Command entry: CHAMFER

Quick Reference

Commands

CHAMFER

Bevels the edges of objects.

System Variables

CHAMFERA

Sets the first chamfer distance when CHAMMODE is set to 0.

CHAMFERB

Sets the second chamfer distance when CHAMMODE is set to 0.

CHAMFERC

Sets the chamfer length when CHAMMODE is set to 1.

CHAMFERD

Sets the chamfer angle when CHAMMODE is set to 1.

CHAMMODE

Sets the input method for CHAMFER

TRIMMODE

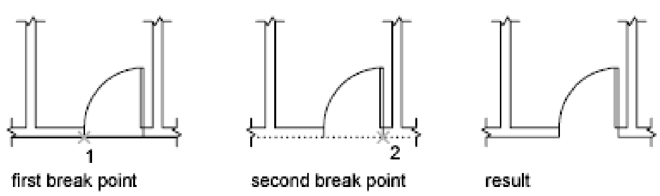
Controls whether selected edges for chamfers and fillets are trimmed.

Break and Join Objects

You can break an object into two objects with or without a gap between them. You can also join objects to make a single object.

Break Objects

Use BREAK to create a gap in an object, resulting in two objects with a gap between them. BREAK is often used to create space for block or text.



To break an object without creating a gap, specify both break points at the same location. The fastest way to do this is to enter @0,0 at the prompt for the second point.

You can create breaks in most geometric objects except

- Blocks
- Dimensions
- Mlines
- Regions

Join Objects

Use JOIN to combine similar objects into a single object. You can also create complete circles and ellipses from arcs and elliptical arcs. You can join

- Arcs
- Elliptical arcs
- Lines
- Polylines
- Splines

The object to which you want to join similar objects is called a source object. Objects to be joined must be located in the same plane. Additional restrictions for each type of objects are described in the JOIN command.

NOTE When joining two or more arcs (or elliptical arcs), the arcs are joined counterclockwise beginning from the source object.

See also:

- Modify or Join Polylines on page 1155
- Modify Complex Objects on page 1153

To break an object

1. Click Home tab ➤ Modify panel ➤ Break.
2. Select the object to break.

By default, the point at which you select the object is the first break point. To select a different pair of break points, enter **f** (First) and specify the first break point.

3 Specify the second break point.

   To break an object without creating a gap, enter **@0,0** to specify the previous point.

Toolbar: Modify 
Command entry: BREAK

To join objects

1 Click Home tab ➤ Modify panel ➤ Join.

2 Select the source object to which you want to join objects.

3 Select one or more objects to join to the source object.

   Valid objects include arcs, elliptical arcs, lines, polylines, and splines. Additional restrictions for each type of objects are described in the JOIN command.

Toolbar: Modify 
Command entry: JOIN

Quick Reference

Commands

BREAK

Breaks the selected object between two points.

JOIN

Joins similar objects to form a single, unbroken object.

Use Grips to Edit Objects

Grips are small squares that are displayed at strategic points on objects that you have selected with a pointing device. You can drag these grips to edit objects directly and quickly.

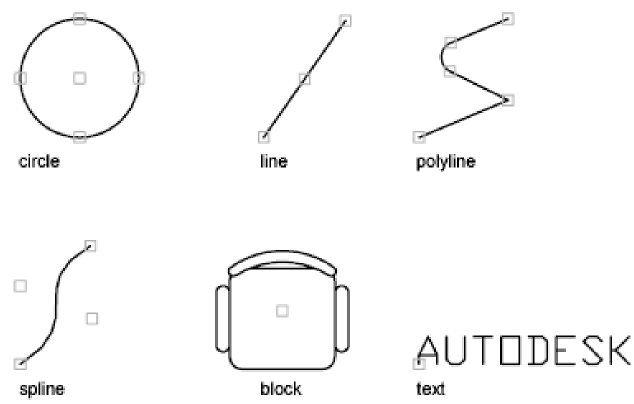
Use Grip Modes

You can drag grips to perform any stretch, move, rotate, scale, or mirror operations. The editing operation you choose to perform is called a *grip mode*.

Grips are small, solid-filled squares that are displayed at strategic points on objects that you have selected with a pointing device. You can drag these grips to stretch, move, rotate, scale, or mirror objects quickly.

When grips are turned on, you can select the objects you want to manipulate *before* entering a command, and then you can manipulate the objects with the pointing device.

NOTE Grips are not displayed on objects that are on locked layers.



To use a grip mode, select a grip (base grip) to act as the base point for the action. (A selected grip is also called a *hot* grip.) Then select one of the grip modes. You can cycle through these modes by pressing Enter or Spacebar. You can also use shortcut keys or right-click to see all of the modes and options.

NOTE When a 2D object lies on a plane other than the current UCS, the object is stretched on the plane on which it was created, not on the plane of the current UCS.

Work with Quadrant Grips

For quadrant grips on circles and ellipses, distance is measured from the center point, not the selected grip. For example, in Stretch mode, you can select a quadrant grip to stretch a circle and then specify a distance at the Command prompt for the new radius. The distance is measured from the center of the circle, not the selected quadrant. If you select the center point to stretch the circle, the circle moves.

Select and Modify Multiple Grips

You can use more than one grip as the base grips for the action. When you select more than one grip (also called *multiple hot grip selection*), the shape of the object is kept intact between the selected grips. To select more than one grip, press and hold the Shift key, and then select the appropriate grips.

Limit the Display of Grips

You can limit the display of grips on selected objects. The GRIPOBJLIMIT system variable suppresses the display of grips when the initial selection set includes more than the specified number of objects. If you add objects to the current selection set, the limit does not apply. For example, if GRIPOBJLIMIT is set to 20, you can select 15 objects and then add 25 objects to the selection, and all objects display grips.

NOTE Grips are not displayed on objects that are on locked layers.

Stretch with Grips

You can stretch an object by moving selected grips to new locations. Grips on text, block references, midpoints of lines, centers of circles, and point objects move the object rather than stretching it. This is an excellent method for moving block references and adjusting dimensions.

Move with Grips

You can move objects by the grip selected. Selected objects are highlighted and are moved the direction and distance of the next point location you specify.

Rotate with Grips

You can rotate selected objects around a base point by dragging and specifying a point location. Alternatively, you can enter an angle value. This is an excellent method for rotating block references.

Scale with Grips

You can scale selected objects relative to a base point. Increase the size of an object by dragging outward from the base grip and specifying a point location, or decrease the size by dragging inward. Alternatively, you can enter a value for relative scaling.

Mirror with Grips

You can mirror selected objects across a temporary mirror line. Turning Ortho on helps you specify a vertical or horizontal mirror line.

See also:

- Use Dynamic Input on page 668
- Work with Dynamic Blocks in Drawings on page 1023

To turn on grips

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Selection tab, select Enable Grips.
3. Click OK.

Command entry: OPTIONS

To set the color of the unselected grip beneath the cursor

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Selection tab, click the arrow under Hover Grip Color.
3. Select a color, or click Select Color to display the Select Color dialog box.

To limit the number of objects in the initial selection set that display grips

1. Click Tools menu ➤ Options.

2 In the Options dialog box, Selection tab, enter a number in the Object Selection Limit for Display of Grips box. The maximum is 32,767.

   If you add objects to the current selection set, the limit does not apply.

To display grip tips for custom objects that support grip tips

1 Click Tools menu ➤ Options. 

2 In the Options dialog box, Selection tab, select Enable Grip Tips.

To cancel grip selection

- Press Esc.

To stretch an object using grips

1 Select the object to stretch.

2 Select a base grip on the object.

   The selected grip is highlighted, and Stretch, the default grip mode, is active.

3 Move the pointing device and click.

   The selected object is stretched as the grip moves.

NOTE To copy the selected object while you're stretching it, press and hold the Ctrl key while you're stretching the object.

To stretch using more than one grip

1 Select several objects to stretch.

2 Hold down Shift and click several grips so that they are highlighted.

3 Release Shift and select a grip as the base grip by clicking the grip.

   The default grip mode, Stretch, is active.

4 Move the pointing device and click.

   The selected grips act in unison and the selected objects are stretched.

To move objects using grips

1. Select the objects to move.
2. Select a base grip on an object by clicking the grip.

   The selected grip is highlighted, and Stretch, the default grip mode, is active.
3. Cycle through the grip modes by pressing Enter until the grip mode Move appears.

   Alternatively, you can right-click to display a shortcut menu of modes and options.
4. Move the pointing device and click.

   The selected objects are moved along with the grip.

NOTE To copy the selected object while you're moving it, press and hold the Ctrl key while you're moving the object.

To rotate objects using grips

1. Select the objects to rotate.
2. Select a base grip on an object by clicking the grip.

   The selected grip is highlighted, and Stretch, the default grip mode, is active.
3. Cycle through the grip modes by pressing Enter until the grip mode Rotate appears.

   Alternatively, you can right-click to display shortcut menu modes and options.
4. Move the pointing device and click.

   The selected objects are rotated around the base grip.

NOTE To copy the selected object while you're rotating it, press and hold the Ctrl key while you're rotating the object.

To scale objects using grips

1. Select the objects to scale.
2. Select a base grip on an object by clicking the grip.

The selected grip is highlighted, and Stretch, the default grip mode, is active.

3 Cycle through the grip modes by pressing Enter until the grip mode Scale appears.

   Alternatively, you can right-click to display shortcut menu modes and options.

4 Enter the scale factor or drag and click to specify a new scale.

NOTE To copy the selected object while you're scaling it, press and hold the Ctrl key while you're scaling the object.

To mirror objects using grips

1 Select the objects to mirror.

2 Select a base grip on an object by clicking the grip.

   The selected grip is highlighted, and the default grip mode, Stretch, is active.

3 Cycle through the grip modes by pressing Enter until the grip mode Mirror appears.

   Alternatively, you can right-click to display shortcut menu modes and options.

4 Click to specify the second point of the mirror line.

   Turning on Ortho mode is often useful when mirroring objects.

Quick Reference

Commands

OPTIONS

Customizes the program settings.

System Variables

GRIPBLOCK

Controls the display of grips in blocks.

GRIPCOLOR

Controls the color of unselected grips.

GRIPHOT

Controls the color of selected grips.

GRIPHOVER

Controls the fill color of an unselected grip when the cursor pauses over it.

GRIPOBJLIMIT

Suppresses the display of grips when the selection set includes more than the specified number of objects.

GRIPS

Controls the use of selection set grips for the Stretch, Move, Rotate, Scale, and Mirror Grip modes.

GRIPSIZE

Sets the size of the grip box in pixels.

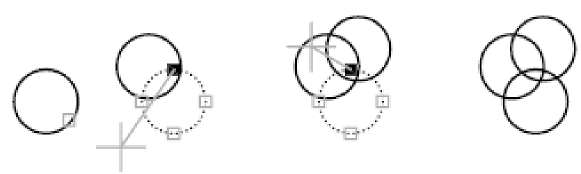
GRIPTIPS

Controls the display of grip tips when the cursor hovers over grips on dynamic blocks and custom objects that support grip tips.

Make Multiple Copies with Grips

You can create multiple copies of objects as you modify them with any of the grip modes.

For example, by using the Copy option, you can rotate the selected objects, leaving copies at each location you specify with the pointing device.

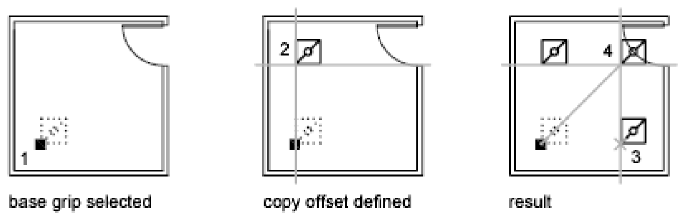


You can also make multiple copies by holding down Ctrl as you select the first point. For example, with the Stretch grip mode, you can stretch an object, such as a line, and then copy it to any point in the drawing area. Multiple copies continue being made until you turn off grips.

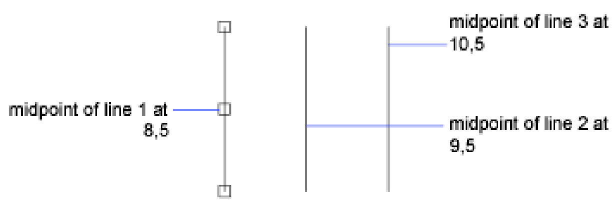
NOTE When you use grips to make multiple copies of an object that contains multiple , only the current scale representation is copied.

Define an Offset Snap or a Rotation Snap

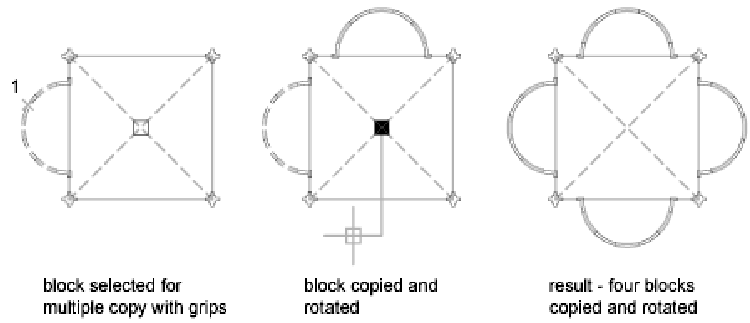
You can place multiple copies at regularly spaced intervals with an offset snap. The offset snap is defined by the distance between an object and the next copy. In the lighting layout below, the first copy of the light fixture symbol is placed at an offset of two units. All subsequent copies are then placed two units apart.



If you hold down Ctrl while you select multiple copy points with the pointing device, the graphics cursor snaps to an offset point based on the last two points you selected. In the illustration below, the midpoint of line 1 is at coordinate 8,5. Based on that midpoint, line 2 was copied using the Ctrl key and Stretch grip mode; its midpoint is at 9,5. The third line snaps to an offset based on the coordinate values 10,5.



Similarly, you can place multiple copies at angular intervals around a base grip with a rotation snap. The rotation snap is defined as the angle between an object and the next copy when you are using Rotate grip mode. Hold down Ctrl to use the rotation snap.



To create copies in any grip mode

1. Select the objects to copy.
2. Select a base grip on an object by clicking the grip.

   The selected grip is highlighted, and the default grip mode, Stretch, is active.
3. Cycle through the grip modes by pressing Enter until the grip mode you want appears.

   Alternatively, you can right-click to display shortcut menu modes and options.
4. Enter **c** (Copy) or press and hold the Ctrl key while you stretch, move, rotate, or scale.

   Copies continue being made until you turn off grips.
5. Enter or specify the additional input required for the current grip mode.
6. Turn off grips by pressing Enter, Spacebar, or Esc.

To create an offset snap for multiple copies using grips

1. Select the objects to copy.
2. Select a base grip on an object by clicking the grip.

   The selected grip is highlighted, and the default grip mode, Stretch, is active.
3. Cycle through the grip modes by pressing Enter until the grip mode Move appears.

   Alternatively, you can right-click to display shortcut menu modes and options.

4. Enter **c** (Copy).
5. Move the cursor and click.

   The offset snap is the distance between the grip you selected and the location you specified for the copy.
6. Hold down Ctrl and place additional copies by specifying additional locations.

   These copies are created at the same offset snap distance as the last copy.
7. Turn off grips by pressing Enter, Spacebar, or Esc.

To create a rotation snap for multiple rotated copies using grips

1. Select the objects to rotate.
2. Select a base grip on an object by clicking the grip.

   The selected grip is highlighted, and the default grip mode, Stretch, is active.
3. Cycle through the grip modes by pressing Enter until the grip mode Rotate appears.

   Alternatively, you can right-click to display shortcut menu modes and options.
4. Enter **c** (Copy).
5. Move the pointing device and click.

   The rotation snap is the angle between the grip you selected and the location you specified for the copy.
6. Hold down Ctrl and place additional copies by specifying additional locations.

   These copies are created at the same rotation snap angle as the first copy.
7. Turn off grips by pressing Enter, Spacebar, or Esc.

To mirror objects and retain the originals using grips

1. Select the objects to mirror.
2. Select a base grip on an object by clicking the grip.

   The selected grip is highlighted, and the default grip mode, Stretch, is active.

3 Cycle through the grip modes by pressing Enter until the grip mode Mirror appears.

   Alternatively, you can right-click to display shortcut menu modes and options.

4 Hold down Ctrl (or enter **c** for Copy) to retain the original image, and specify the second point of the mirror line.

   Turning on Ortho mode is often useful when mirroring objects.

5 Turn off grips by pressing Enter, Spacebar, or Esc.

Quick Reference

Commands

OPTIONS

Customizes the program settings.

System Variables

GRIPBLOCK

Controls the display of grips in blocks.

GRIPCOLOR

Controls the color of unselected grips.

GRIPHOT

Controls the color of selected grips.

GRIPS

Controls the use of selection set grips for the Stretch, Move, Rotate, Scale, and Mirror Grip modes.

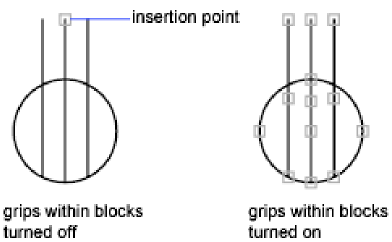
GRIPSIZE

Sets the size of the grip box in pixels.

Control Grips in Blocks

You can specify whether a block displays a single grip or multiple grips.

You can specify whether a selected block reference displays a single grip at its insertion point or displays multiple grips associated with the objects grouped within the block.



See also:

- Specify Grips for Dynamic Blocks on page 987

To turn grips within blocks on or off

1. Click Tools menu ➤ Options. 
2. In the Options dialog box, Selection tab, select or clear Enable Grips Within Blocks.
3. Click OK.

Command entry: OPTIONS

Quick Reference

Commands

OPTIONS

Customizes the program settings.

System Variables

GRIPBLOCK

Controls the display of grips in blocks.

GRIPCOLOR

Controls the color of unselected grips.

GRIPHOT

Controls the color of selected grips.

GRIPS

Controls the use of selection set grips for the Stretch, Move, Rotate, Scale, and Mirror Grip modes.

GRIPSIZE

Sets the size of the grip box in pixels.

Modify Complex Objects

Additional editing operations are available for complex objects, such as blocks, dimensions, hatches, and polylines.

Disassociate Compound Objects (Explode)

You can convert a compound object, such as a polyline, dimension, hatch, or block reference, into individual elements.

You can explode a compound object, such as a polyline, dimension, hatch, or block reference, to convert it into individual elements. For example, exploding a polyline breaks it down to simple lines and arcs. Exploding a block reference or an associative dimension replaces it with copies of the objects that compose the block or dimension.

Explode Dimensions and Hatches

When you explode a dimension or a hatch, all associativity is lost and the dimension or hatch object is replaced by individual objects such as lines, text, points, and 2D solids. To explode dimensions automatically when you create them, set the DIMASSOC system variable to 0.

Explode Polylines

When you explode a polyline, any associated width information is discarded. The resulting lines and arcs follow the polyline's centerline. If you explode a block that contains a polyline, you need to explode the polyline separately. If you explode a donut, its width becomes 0.

Explode Block References

If you explode a block with attributes, the attribute values are lost, leaving only the attribute definitions. The colors and linetypes of objects in exploded block references can change.

Explode External References

An external reference (xref) is a drawing file linked (or attached) to another drawing. You cannot explode xrefs and their dependent blocks.

To explode an object

1. Click Home tab ➤ Modify panel ➤ Explode.
2. Select the objects to be exploded.

   For most objects, exploding has no visible effect.

Toolbar: Modify
Command entry: EXPLODE

Quick Reference

Commands

EXPLODE

Breaks a compound object into its component objects.

XPLODE

Breaks a compound object into its component objects.

System Variables

DIMASSOC

Controls the associativity of dimension objects and whether dimensions are exploded.

EXPLMODE

Controls whether the EXPLODE command supports nonuniformly scaled (NUS) blocks.

Modify or Join Polylines

Additional editing operations are available for changing the shape and display of polyline objects. You can also join separate polylines.

You can modify polylines in several ways using PEDIT, the Properties palette, or grips including the following:

- Move, add, or delete individual vertices
- Set a uniform width for the entire polyline or control the width of each segment
- Create an approximation of a spline called a *spline-fit polyline*
- Display noncontinuous linetypes with or without a dash before and after each vertex
- Change the orientation of text in a linetype assigned to a polyline by reversing its direction

Join Polyline Segments

You can join a line, an arc, or another polyline to an open polyline if their ends connect or are close to each other.

If the ends are not coincident but are within a distance that you can set, called the *fuzz distance*, the ends are joined by either trimming them, extending them, or connecting them with a new segment.

Spline-fit polylines return to their original shape when joined. Polylines cannot be joined into a Y shape.

If the properties of several objects being joined into a polyline differ, the resulting polyline inherits the properties of the first object that you selected.

Modify a Segment Within a Polyline

You can modify a segment in a polyline using the MOVE, ROTATE, or SCALE commands. You select the single arc or line segment, also called a *subobject*, within a polyline by pressing Ctrl when you click it.

You can also use the Stretch, Move, Rotate, and Scale grip modes to modify the polyline segment. Properties of polyline segments such as color or linetype, cannot be modified individually, but you can use the Properties palette to modify the width of individual segments.

See also:

- Trim or Extend Objects on page 1116
- Break and Join Objects on page 1138

To modify a polyline

1. Click Home tab ➤ Modify panel ➤ Edit Polyline.
2. Select the polyline to modify.

   NOTE You can select a single arc or line segment, also called a subobject, within a polyline by pressing the Ctrl key when you click it.

   ---

3. If the selected object is a spline, 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.

   Before the selected spline is converted to a polyline, the following prompt is displayed:

   Specify a precision <10>: *Enter a new precision value or press* Enter

   The PLINECONVERTMODE system variable determines whether the polylines are created with linear or arc segments. When the PEDITACCEPT system variable is set to 1, this prompt is suppressed, and the selected object is automatically converted to a polyline.

4. Edit the polyline by entering one or more of the following options:
   - Enter **c** (Close) to create a closed polyline.
   - Enter **j** (Join) to join contiguous lines, splines, arcs, or polylines.
   - Enter **w** (Width) to specify a new uniform width for the entire polyline.
   - Enter **e** (Edit Vertex) to edit a vertex.
   - Enter **f** (Fit) to create a series of arcs joining each pair of vertices.
   - Enter **s** (Spline) to create an approximation of a spline.
   - Enter **d** (Decurve) to remove extra vertices inserted by a fit or spline curve and to straighten all segments of the polyline.

- Enter **L** (Ltype Gen) to generate the linetype in a continuous pattern through the vertices of the polyline.
- Enter **r** (Reverse) to reverse the order of vertices of the polyline.
- Enter **u** (Undo) to reverse actions back to the start of PEDIT.

5 Enter **x** (Exit) to end a command option. Press Enter to exit the PEDIT command.

Toolbar: Modify II
Command entry: PEDIT

To reverse lines, polylines, splines, or helixes

1 Click Home tab ➤ Modify panel ➤ Reverse.
2 Select a line, polyline, spline, or helix to reverse.
3 Press Enter to end the command.

Command entry: REVERSE

To join polylines, splines, lines, and arcs into a single polyline

1 Click Home tab ➤ Modify panel ➤ Edit Polyline.
2 Select a polyline, spline, line, or arc to edit. If you selected a spline, line, or arc, press Enter to convert the selected object into a polyline.
3 Enter **j** (Join).
4 Select one or more polylines, splines, lines, or arcs that are located end to end.

   Each selected polyline, spline, line, or arc is now joined into a single polyline.
5 Press Enter to end the command.

Toolbar: Modify II
Command entry: PEDIT

To delete a vertex in a polyline

1 Click Home tab ➤ Modify panel ➤ Edit Polyline.

2 Select a polyline.

3 Enter **e** (Edit vertex).

   The first vertex is marked with an X. Use the Next option to move the X to the vertex preceding the one that you want to delete.

4 Enter **s** (Straighten).

5 Use the Next option to move the X to the vertex immediately following the one that you want to delete.

6 Enter **g** (Go).

   The vertex on the polyline is deleted. The vertices on either side of the deleted vertex are joined by a straight polyline segment.

7 Enter **x** (Exit) to end editing vertices.

8 Press Enter to end the command.

Toolbar: Modify II
Command entry: PEDIT

To taper the width of individual polyline segments

1 Click Home tab ➤ Modify panel ➤ Edit Polyline.

2 Select the polyline to edit.

3 Enter **e** (Edit Vertex).

   The first vertex is marked with an X. Move to the appropriate vertex with Next or Previous.

4 Enter **w** (Width).

5 Enter new starting and ending widths, and press Enter to move to the next vertex. Repeat steps 4 and 5 for each segment.

6 Enter **u** (Undo) to reverse actions back to the start of PEDIT.

7 Enter **x** (Exit) to end editing vertices.

8 Press Enter to end the command.

Toolbar: Modify II

Command entry: PEDIT

Quick Reference

Commands

PEDIT

Edits polylines and 3D polygon meshes.

JOIN

Joins similar objects to form a single, unbroken object.

REVERSE

Reverses the order of vertices of the selected lines, polylines, splines, and helixes.

System Variables

PEDITACCEPT

Suppresses display of the Object Selected Is Not a Polyline prompt in PEDIT.

PLINECONVERTMODE

Specifies the fit method used in converting splines to polylines.

SPLFRAME

Controls the display of splines and spline-fit polylines.

SPLINESEGS

Sets the number of line segments to be generated for each spline-fit polyline generated by the Spline option of the PEDIT command.

SPLINETYPE

Sets the type of curve generated by the Spline option of the PEDIT command.

SURFTYPE

Controls the type of surface-fitting to be performed by the Smooth option of the PEDIT command.

SURFU

Sets the surface density for PEDIT Smooth in the M direction and the U isolines density on surface objects.

SURFV

Sets the surface density for PEDIT Smooth in the N direction and the V isolines density on surface objects.

Modify Splines

Additional editing options are available for changing the shape of spline objects.

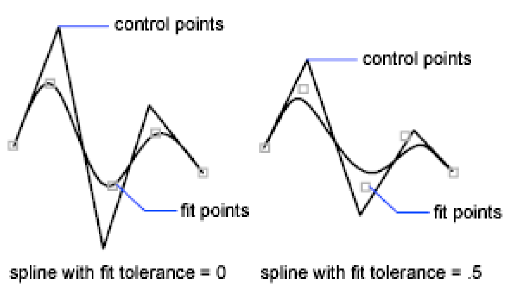
In addition to the general editing operations available for most objects, there are additional options available for editing splines with SPLINEDIT.

You also can change the tolerance of the spline. Tolerance refers to how closely the spline fits the set of fit points you specify. The lower the tolerance, the more closely the spline fits the points.

Edit Splines with Grips

When you select a spline, grips are displayed on its fit points (the GRIPS system variable must be set to 1). You can use grips to modify the shape and location of the spline.

After certain operations, fit points are discarded and grips are displayed on control points instead. These operations include trimming the spline, moving the control points, and purging the fit data. If the spline's control frame is turned on (the SPLFRAME system variable is set to 1), grips are displayed on both the spline's control points and its fit points, when available.

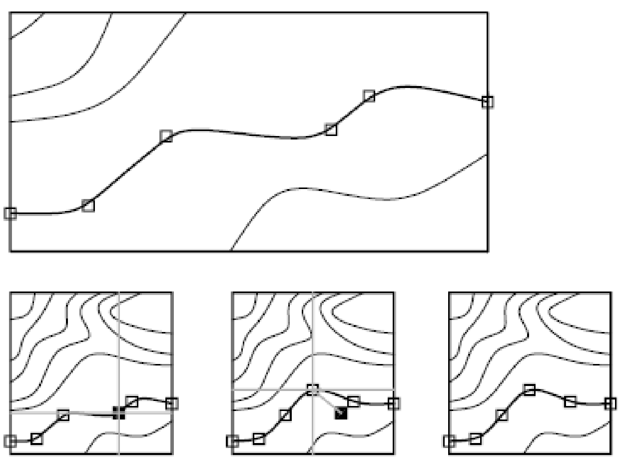


You can delete fit points of a spline, add fit points for greater accuracy, or move fit points to alter the shape of a spline. You can open or close a spline and edit the spline start and end tangents. Spline direction is reversible. You can change the *tolerance* of the spline also. Tolerance refers to how closely the spline fits the set of fit points you specify. The lower the tolerance, the more closely the spline fits the points.

Refine the Shape of a Spline

You can refine a spline by increasing the number of control points in one portion of the spline or by changing the weight of specific control points. Increasing the weight of a control point pulls the spline more towards that point. You can also refine a spline by changing its order. A spline's order is the degree of the spline polynomial + 1. A cubic spline, for example, has order 4. The higher a spline's order, the more control points it has.

Consider the following example. You have created a spline to represent a geographic contour. Grips are turned on, and you need to move the fourth fit point to increase accuracy. When you select the spline, grips appear at the control points. If you created the spline by fitting it through a set of points, and you have not purged this information using the Purge option of the SPLINEDIT command, and you select the Fit Data option, grips appear at the fit points on the selected spline instead of at the control points.



fourth fit point selected | fit point moved | result

See also:

- Break and Join Objects on page 1138

To edit a spline

1 Click Home tab ➤ Modify panel ➤ Edit Spline.

2 Select the spline to modify.

3 Edit the spline by entering one or more of the following options:
   - Enter **f** (Fit) to edit the fit data that defines the spline.
   - Enter **c** (Close) to change an open spline into a continuous, closed loop.
   - Enter **m** (Move Vertex) to move a fit point to a new location.
   - Enter **r** (Refine) to modify a spline definition by adding and weighting control points and elevating the order of the spline.
   - Enter **e** (Reverse) to reverse the direction of the spline.
   - Enter **p** (Convert to Polyline) to convert the spline to polyline.
   - Enter **u** (Undo) to cancel the last editing action.

4 Enter **x** (Exit) to end the command.

Toolbar: Modify II 
Command entry: SPLINEDIT

To convert a spline to a polyline

1 Click Home tab ➤ Modify panel ➤ Edit Spline.

2 Select the spline to convert.

3 Select the Convert to Polyline option.

4 Specify a precision value or press Enter to end the command.

Toolbar: Modify II 
Command entry: SPLINEDIT

Quick Reference

Commands

JOIN

Joins similar objects to form a single, unbroken object.

SPLINEDIT

Edits a spline or spline-fit polyline.

REVERSE

Reverses the order of vertices of the selected lines, polylines, splines, and helixes.

System Variables

PLINECONVERTMODE

Specifies the fit method used in converting splines to polylines.

SPLFRAME

Controls the display of splines and spline-fit polylines.

Modify Helixes

You can use grips or the Properties palette to modify the shape and size of a helix.

You can use the grips on a helix to change the following properties:

- Start point
- Base radius
- Top radius
- Height
- Location

When you use a grip to change the base radius of a helix, the top radius scales to maintain the current ratio. Use the Properties palette to change the base radius independent of the top radius.

You can use the Properties palette to change other helix properties, such as

- Number of turns (Turns)
- Turn height
- Direction of the twist — clockwise (CW) or counterclockwise (CCW)

With the Constrain property, you can specify that the Height, Turns, or Turn Height properties of the helix are constrained. The Constrain property affects how the helix changes when the Height, Turns, or Turn Height properties are changed either through the Properties palette or through grip editing. The table below shows the behavior of the helix depending on which property is constrained.

Constrained property	Property to change	Effect on these helix properties		
		Height	**Turns**	**Turn Height**
Height	Height	Changed	Fixed	Changed
	Turns	Fixed	Changed	Changed
	Turn Height	Fixed	Changed	Changed
Turns	Height	Changed	Fixed	Changed
	Turns	Fixed	Changed	Changed
	Turn Height	Changed	Fixed	Changed
Turn Height	Height	Changed	Changed	Fixed
	Turns	Changed	Changed	Fixed
	Turn Height	Fixed	Changed	Changed

See also:

- Draw Helixes on page 839

Quick Reference

Commands

HELIX

Creates a 2D spiral or 3D spring.

Modify Multilines

Multiline objects are composed of 1 to 16 parallel lines, called *elements*. To modify multilines or their elements, you can use common editing commands, a multiline editing command, and multiline styles.

Special multiline editing features are available with the MLEDITcommand including the following:

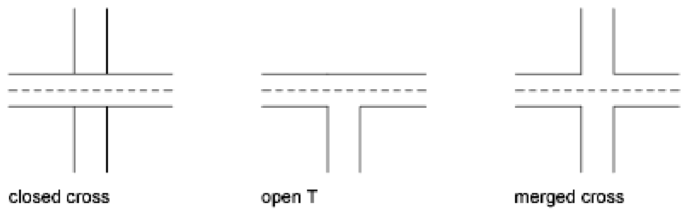
- Add or delete a vertex
- Control the visibility of corner joints
- Control the style of intersection with other multilines
- Open or close gaps in a multiline object

Add and Delete Multiline Vertices

You can add or delete any vertex in a multiline.

Edit Multiline Intersections

If you have two multilines in a drawing, you can control the way they intersect. Multilines can intersect in a cross or a **T** shape, and the crosses or **T** shapes can be closed, open, or merged.



Edit Multiline Styles

You can use MLSTYLE to edit multiline styles to change the properties of multiline elements or the end caps and background fill of subsequently created multilines.

Multiline styles control the number of line elements in a multiline and the color, linetype, lineweight, and offset of each element. You can also modify the display of joints, end caps, and background fill.

Multiline styles have the following limitations:

- You cannot edit the element and multiline properties of the STANDARD multiline style or any multiline style already used in the drawing.
- To edit an existing multiline style, you must do so *before* you draw any multilines in that style.

NOTE If you use MLSTYLE to create a multiline style without saving it, and then select another style or create a new style, the first MLSTYLE properties are lost. To maintain the properties, save each multiline style to an MLN file before creating a new one.

Use Common Editing Commands on Multilines

You can use most of the common editing commands on multilines *except*

- BREAK
- CHAMFER
- FILLET
- LENGTHEN
- OFFSET

To perform these operations, first use EXPLODE to replace the multiline object with separate line objects.

NOTE If you trim or extend a multiline object, only the first boundary object encountered determines the shape of the end of the multiline. A multiline cannot have a complex boundary at its endpoint.

See also:

- Draw Multiline Objects on page 810

To delete a vertex from a multiline

1 Click Modify menu ➤ Object ➤ Multiline.

2 In the Multiline Edit Tools dialog box, select Delete Vertex.

3 In the drawing, specify the vertex to delete. Press Enter.

Command entry: MLEDIT

To create a closed cross intersection

1 Click Modify menu ➤ Object ➤ Multiline.

2 In the Multiline Edit Tools dialog box, select Closed Cross.

3 Select the multiline for the foreground.

4 Select the multiline for the background.

   The intersection is modified. You can continue selecting intersecting multilines to modify, or press Enter to end the command. Press Enter again to redisplay the Multiline Edit Tools dialog box.

To edit a multiline style

1 Click Format menu ➤ Multiline Style. 

2 In the Multiline Styles dialog box, select the style name from the list. Click Modify.

3 Click Element Properties.

4 In the Modify Multiline Styles dialog box, change the settings as needed.

5 Click OK.

6 In the Multiline Styles dialog box, click Save to save the changes to the style in the MLN file.

7 Click OK.

Command entry: MLSTYLE

Quick Reference

Commands

MLEDIT

Edits multiline intersections, breaks, and vertices.

MLSTYLE

Creates, modifies, and manages multiline styles.

Work with 3D Models

Create 3D Models

20

Use three-dimensional models to help you visualize and test your designs in 3D space.

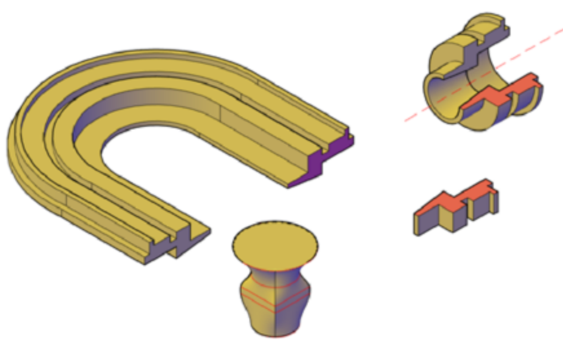
Overview of 3D Modeling

With 3D modeling, you can design using solid, surface, and mesh models.

Create new 3D solids and surfaces, or sweep, combine, and modify existing objects. Create or convert objects to mesh to obtain enhanced smoothing and creasing capabilities. You can also use simulated surfaces (3D thickness), or wireframe models to represent 3D objects.

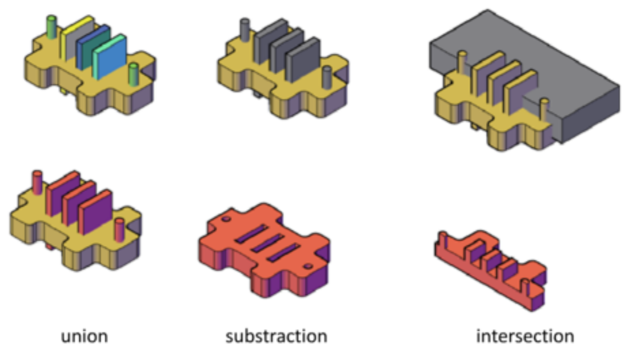
Solid Models

A solid model is a 3D representation that has such properties as mass, volume, center of gravity, and moments of inertia.



Solid models contain the most information and are the least ambiguous of the 3D modeling types. You can analyze solids for their mass properties and export data to applications that do NC (numerical control) milling or FEM (finite element method) analysis.

Use solid models as the building blocks for your model. You can start with primitive solids such as cones, boxes, cylinders, and pyramids. Draw a custom polysolid extrusion or use various sweeping operations to create solids whose shapes conform to a path you specify. Then modify or recombine objects to create new solid shapes.

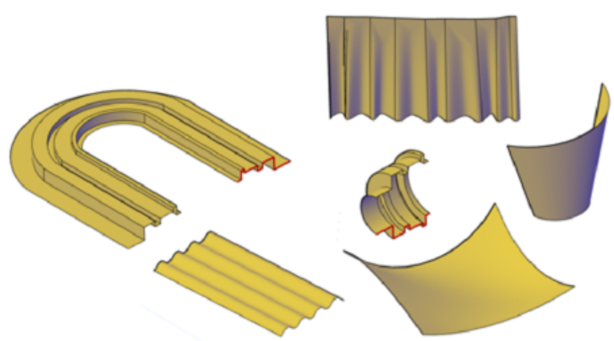


Complex solid shapes are easier to construct and edit than legacy wireframe models. However, if needed, you can explode a solid to break it down to regions, bodies, surfaces, and wireframe objects.

NOTE When you work with 3D solids, you occasionally see messages that refer to *ASM* or *ShapeManager*. ShapeManager® is the Autodesk technology that provides 3D solid modeling capabilities to AutoCAD and other products.

Surface Model

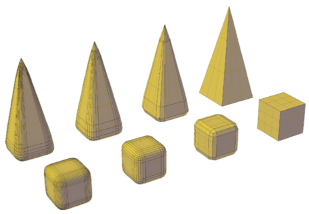
A surface model represents an infinitely thin shell that corresponds to the shape of a 3D object.



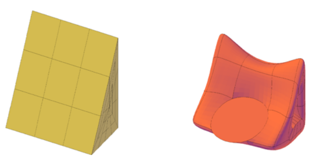
You create surface models using some of the same tools that you use for solid models. For example, you can use sweeping, lofting, and revolving to create a surface model. The difference is that surface models are open ended. Solid models are closed.

Mesh Model

A mesh model consists of vertices, edges, and faces that use polygonal representation (including triangles and quads) to define a 3D shape.



Unlike solid models, mesh has no mass properties. However, as with 3D solids, you can create primitive mesh forms such as boxes, cones, and pyramids, starting in AutoCAD 2010. You can then modify mesh models in ways that are not available for 3D solids or surfaces. For example you can apply creases, splits, and increasing levels of smoothness. You can drag mesh subobjects (faces, edges, and vertices) to deform the object. To achieve more granular results, you can refine the mesh in specific areas before modifying it.



Use mesh models to provide the hiding, shading, and rendering capabilities of a solid model without the physical properties such as mass, moments of inertia, and so on.

Advantages of 3D Modeling

Modeling in 3D has several advantages. You can

- View the model from any vantage point
- Generate reliable standard and auxiliary 2D views automatically
- Create sections and 2D drawings
- Remove hidden lines and do realistic shading
- Check interferences and perform engineering analysis

- Add lighting and create realistic rendering
- Navigate through the model
- Use the model to create an animation
- Extract manufacturing data

See also:

- Enter 3D Coordinates on page 637
- Specify Workplanes in 3D (UCS) on page 650
- Use the Dynamic UCS with Solid Models on page 660

Quick Reference

Commands

BOX

Creates a 3D solid box.

CONE

Creates a 3D solid cone.

CYLINDER

Creates a 3D solid cylinder.

EXTRUDE

Extends the dimensions of a 2D object or 3D face into 3D space.

LOFT

Creates a 3D solid or surface in the space between several cross sections.

MESH

Creates a 3D mesh primitive object such as a box, cone, cylinder, pyramid, sphere, wedge, or torus.

POLYSOLID

Creates a 3D wall-like polysolid.

PYRAMID

Creates a 3D solid pyramid.

REVOLVE

Creates a 3D solid or surface by sweeping a 2D object around an axis.

SPHERE

Creates a 3D solid sphere.

SWEEP

Creates a 3D solid or surface by sweeping a 2D object along a path.

TORUS

Creates a donut-shaped 3D solid.

WEDGE

Creates a 3D solid wedge.

System Variables

DELOBJ

Controls whether geometry used to create 3D objects is retained or deleted.

PSOLWIDTH

Controls the default width for a swept solid object created with the POLYSOLID command.

PSOLHEIGHT

Controls the default height for a swept solid object created with the POLYSOLID command.

Create 3D Solids and Surfaces

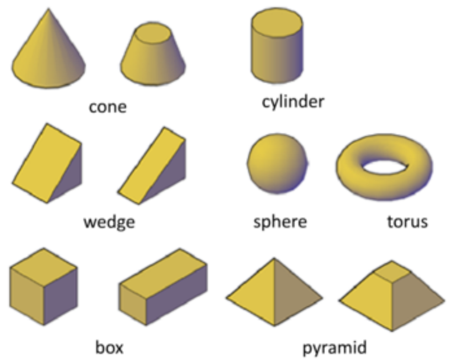
Create 3D solids and surfaces from primitives or by combining or extending existing objects.

Overview of Creating 3D Solids and Surfaces

Three-dimensional solid objects often start as one of several basic shapes, or primitives, that you can then modify and recombine. A 3D solid or surface can also be the result of extruding a 2D shape to follow a specified path in 3D space.

About Solid Primitives

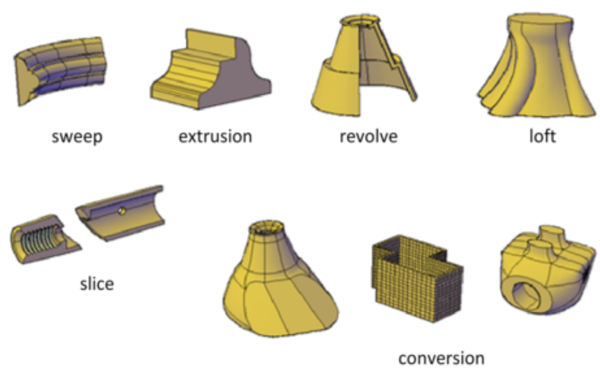
You can create several basic 3D shapes, known as *solid primitives*: boxes, cones, cylinders, spheres, wedges, pyramids, and tori (donuts).



By combining primitive shapes, you can create more complex solids. For example, you can join two solids, subtract one from the other, or create a shape based on the intersection of their volumes.

About Solids Based on Other Objects

You can also create 3D solids and surfaces from existing objects.



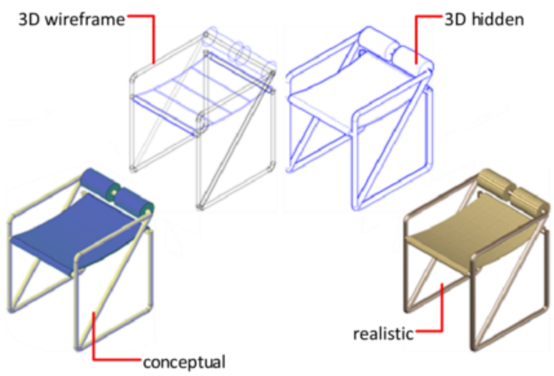
The following methods are available:

- **Sweep.** Extends a 2D object along a path.

- **Extrusion.** Extends the shape of a 2D object in a perpendicular direction into 3D space.
- **Revolve.** Sweeps a 2D object around an axis.
- **Loft.** Extends the contours of a shape between one or more open or closed objects.
- **Slice.** Divides a solid object into two separate 3D objects.
- **Conversion.** Converts mesh objects and planar objects with thickness into solids and surfaces.

Use Visual Styles with 3D Objects

Solids and surfaces can be displayed in one of several visual styles that are applied to the viewport.



Quick Reference

Commands

BOX

Creates a 3D solid box.

CONE

Creates a 3D solid cone.

CONVTOSOLID

Converts 3D meshes and polylines and circles with thickness to 3D solids.

CONVTOSURFACE

Converts objects to 3D surfaces.

CYLINDER

Creates a 3D solid cylinder.

EXTRUDE

Extends the dimensions of a 2D object or 3D face into 3D space.

INTERSECT

Creates a 3D solid, surface, or 2D region from overlapping solids, surfaces, or regions.

INSERT

Inserts a block or drawing into the current drawing.

LOFT

Creates a 3D solid or surface in the space between several cross sections.

PYRAMID

Creates a 3D solid pyramid.

REVOLVE

Creates a 3D solid or surface by sweeping a 2D object around an axis.

SLICE

Creates new 3D solids and surfaces by slicing, or dividing, existing objects.

SUBTRACT

Combines selected 3D solids, surfaces, or 2D regions by subtraction.

SWEEP

Creates a 3D solid or surface by sweeping a 2D object along a path.

TORUS

Creates a donut-shaped 3D solid.

UNION

Combines selected 3D solids, surfaces, or 2D regions by addition.

VISUALSTYLES

Creates and modifies visual styles and applies a visual style to a viewport.

WEDGE

Creates a 3D solid wedge.

System Variables

FACETRES

Adjusts the smoothness of shaded and rendered objects and objects with hidden lines removed.

ISOLINES

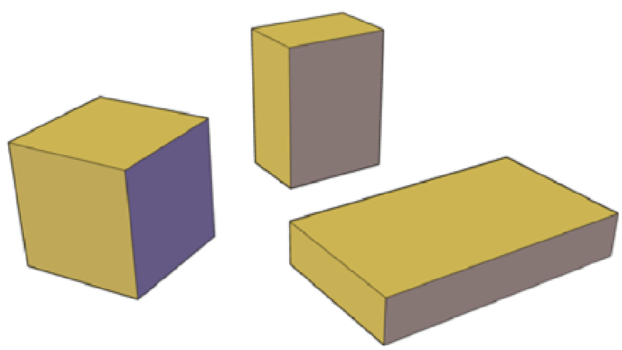
Specifies the number of contour lines per surface on objects.

Create 3D Solid Primitives

Start with standard shapes known as *solid primitives* to create boxes, cones, cylinders, spheres, tori (donuts), wedges, and pyramids.

Create a Solid Box

Create a rectangular or cubical solid box.



The base of the box is always drawn parallel to the *XY* plane of the current UCS (workplane).

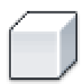
Box Creation Options

Use the following options to control the size and rotation of the boxes you create:

- **Create a cube.** Use the Cube option of the BOX command to create a box with sides of equal length.
- **Specify rotation.** Use the Cube or Length option if you want to set the rotation of the box in the *XY* plane.
- **Start from the center point.** Use the Center Point option to create a box using a specified center point.

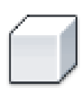
To create a solid box based on two points and a height

1. Click Home tab ➤ Modeling panel ➤ Box.
2. Specify the first corner of the base.
3. Specify the opposite corner of the base.
4. Specify the height.

Toolbar: Modeling 
Command entry: BOX

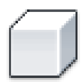
To create a solid box based on length, width, and height

1. Click Home tab ➤ Modeling panel ➤ Box.
2. Specify the first corner of the base.
3. At the Command prompt, enter **l** (Length). Specify the length of the base.
4. Specify the width of the base.
5. Specify the height.

Toolbar: Modeling 
Command entry: BOX

To create a solid box based on a center point, corner of base, and height

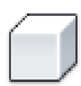
1. Click Home tab ➤ Modeling panel ➤ Box.
2. At the Command prompt, enter **c** (Center). Specify the center point of the base.
3. To set the location of a corner of the base, use one of the following methods:
   - To set the length and width simultaneously: Specify the location of one corner of the base.
   - To set the length and width separately: At the Command prompt, enter **l** (Length) and specify the length. Then specify the width.
4. Specify the height.

Toolbar: Modeling 
Command entry: BOX

To create a solid cube

1. Click Home tab ➤ Modeling panel ➤ Box.
2. Specify the first corner, or enter **c** (Center) to specify the center point of the base.
3. At the Command prompt, enter **c** (Cube). Specify the length of the cube and a rotation angle.

   The length value sets both the width and height of the cube.

Toolbar: Modeling 
Command entry: BOX

Quick Reference

Commands

BOX

Creates a 3D solid box.

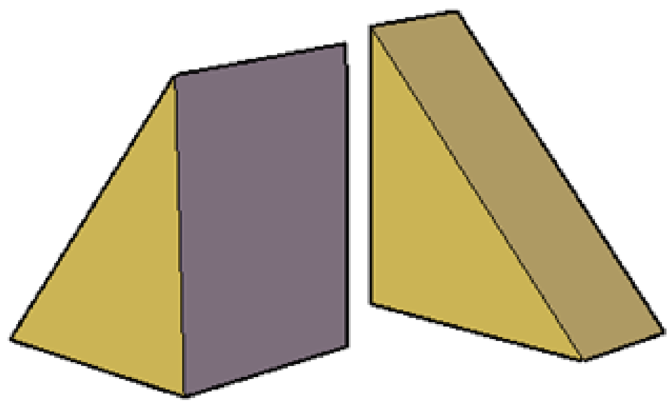
System Variables

DRAGVS

Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Create a Solid Wedge

Create a solid wedge with rectangular or cubical faces.

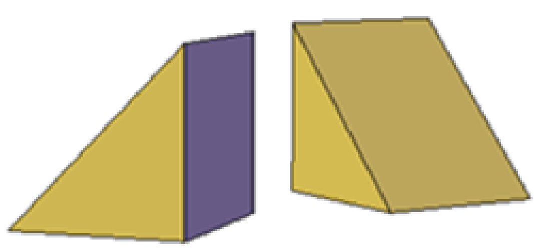


The base of the wedge is drawn parallel to the *XY* plane of the current UCS with the sloped face opposite the first corner. The height of the wedge is parallel to the *Z* axis.

Wedge Creation Options

Use the following options to control the size and rotation of the wedges you create:

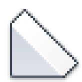
- **Create a wedge with sides of equal length.** Use the Cube option of the WEDGE command.



- **Specify rotation.** Use the Cube or Length option if you want to set the rotation of the wedge in the *XY* plane.
- **Start from the center point.** Use the Center Point option to create a wedge using a specified center point.

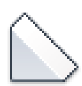
To create a solid wedge based on two points and a height

1. Click Home tab ➤ Modeling panel ➤ Wedge.
2. Specify the first corner of the base.
3. Specify the opposite corner of the base.
4. Specify the height of the wedge.

Toolbar: Modeling 
Command entry: WEDGE

To create a solid wedge based on length, width, and height

1. Click Home tab ➤ Modeling panel ➤ Wedge.
2. Specify the first corner of the base.
3. At the Command prompt, enter **l** (Length). Specify the length of the base.
4. Specify the width of the base.
5. Specify the height of the wedge.

Toolbar: Modeling 
Command entry: WEDGE

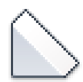
To create a solid wedge based on a center point, corner of base, and height

1. Click Home tab ➤ Modeling panel ➤ Wedge.
2. At the Command prompt, enter **c** (Center). Specify the center point of the base.

3 To set the location of a corner of the base, use one of the following methods:

- *To set the length and width simultaneously:* Specify the location of one corner of the base.
- *To set the length and width separately:* At the Command prompt, enter **l** (Length) and specify the length. Then specify the width.

4 Specify the height of the wedge.

Toolbar: Modeling 
Command entry: WEDGE

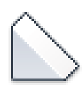
To create a solid wedge with equal length, width, and height

1 Click Home tab ➤ Modeling panel ➤ Wedge.

2 Specify the first corner or enter **c** (Center) to set the center point of the base.

3 At the Command prompt, enter **c** (Cube). Specify the length of the wedge and a rotation angle.

The length value sets both the width and height of the wedge.

Toolbar: Modeling 
Command entry: WEDGE

Quick Reference

Commands

WEDGE

Creates a 3D solid wedge.

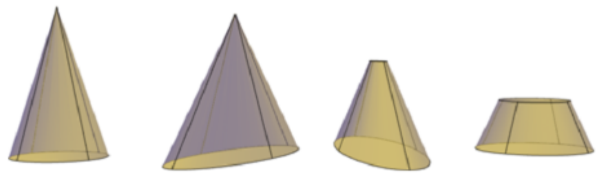
System Variables

DRAGVS

Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Create a Solid Cone

Create a pointed or frustum of a cone with a circular or elliptical base.

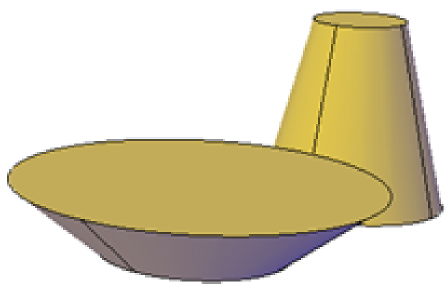


By default, the base of the cone lies on the *XY* plane of the current UCS. The height of the cone is parallel to the *Z* axis.

Cone Creation Options

Use the following options to control the size and rotation of the cones you create:

- **Set the height and orientation.** Use the Axis Endpoint option of the CONE command. Use the Top Radius option to specify the axis endpoint as the point of the cone or the center of the top face. The axis endpoint can be located anywhere in 3D space.
- **Create a frustum of a cone.** Use the Top Radius option of the CONE command to create a frustum, which tapers to an elliptical or planar face.

  The Frustum tool is also available from the Modeling tab of the tool palette. You can also use grips to modify the tip of a cone and convert it to a flat face.
- **Specify circumference and base plane.** The 3P (Three Points) option of the CONE command defines the size and plane of the base of the cone anywhere in 3D space.
- **Define the angle of the taper.** To create a conical solid that requires a specific angle to define its sides, draw a 2D circle. Then use EXTRUDE and

the Taper Angle option to taper the circle at an angle along the *Z* axis. This method, however, creates an extruded solid, not a true solid cone primitive.

See also:

- Use Grips to Edit 3D Solids and Surfaces on page 1291

To create a solid cone with a circular base

1 Click Home tab ➤ Modeling panel ➤ Cone.

2 Specify the center point of the base.

3 Specify the radius or diameter of the base.

4 Specify the height of the cone.

Toolbar: Modeling 
Command entry: CONE

To create a solid cone with an elliptical base

1 Click Home tab ➤ Modeling panel ➤ Cone.

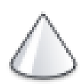
2 At the Command prompt, enter **e** (Elliptical).

3 Specify the start point of first axis.

4 Specify the endpoint of the first axis.

5 Specify the endpoint (length and rotation) of the second axis.

6 Specify the height of the cone.

Toolbar: Modeling 
Command entry: CONE

To create a frustum of a solid cone

1 Click Home tab ➤ Modeling panel ➤ Cone.

2 Specify the center point of the base.

3 Specify the radius or diameter of the base.

4 At the Command prompt, enter **t** (Top radius). Specify the top radius.

5 Specify the height of the cone.

Toolbar: Modeling 

Command entry: CONE

To create a solid cone with the height and orientation specified by the axis endpoint

1 Click Home tab ➤ Modeling panel ➤ Cone.

2 Specify the center point of the base.

3 Specify the radius or diameter of the base.

4 At the Command prompt, enter **a** (Axis endpoint). Specify the endpoint and rotation of the cone.

This endpoint can be located anywhere in 3D space.

Toolbar: Modeling 

Command entry: CONE

Quick Reference

Commands

CONE

Creates a 3D solid cone.

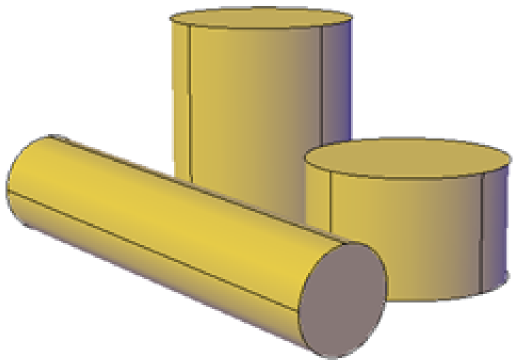
System Variables

DRAGVS

Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Create a Solid Cylinder

Create a solid cylinder with a circular or elliptical base.



By default, the base of the cylinder lies on the *XY* plane of the current UCS. The height of the cylinder is parallel to the *Z* axis.

Cylinder Creation Options

Use the following options to control the size and rotation of the cylinders you create:

- **Set rotation.** Use the Axis Endpoint option of the CYLINDER command to set the height and rotation of the cylinder. The center point of the top plane of the cylinder is the axis endpoint, which can be located anywhere in 3D space.
- **Use three points to define the base.** Use the 3P (Three Points) option to define the base of the cylinder. You can set three points anywhere in 3D space.
- **Construct a cylindrical form with special detail, such as grooves.** Create a closed polyline (PLINE to represent a 2D profile of the base. Use EXTRUDE to define the height along the *Z* axis. The resulting extruded solid is not a true solid cylinder primitive.

To create a solid cylinder with a circular base

1. Click Home tab ➤ Modeling panel ➤ Cylinder.
2. Specify the center point of the base.
3. Specify the radius or diameter of the base.
4. Specify the height of the cylinder.

Toolbar: Modeling
Command entry: CYLINDER

To create a solid cylinder with an elliptical base

1. Click Home tab ➤ Modeling panel ➤ Cylinder.
2. At the Command prompt, enter **e** (Elliptical).
3. Specify the start point of the first axis.
4. Specify the endpoint of the first axis.
5. Specify the endpoint (length and rotation) of the second axis.
6. Specify the height of the cylinder.

Toolbar: Modeling
Command entry: CYLINDER

To create a solid cylinder with the height and rotation specified (axis endpoint)

1. Click Home tab ➤ Modeling panel ➤ Cylinder.
2. Specify the center point of the base.
3. Specify the radius or diameter of the base.
4. At the Command prompt, enter **a** (Axis endpoint). Specify the axis endpoint of the cylinder.

   This endpoint can be located anywhere in 3D space.

Toolbar: Modeling
Command entry: CYLINDER

Quick Reference

Commands

CYLINDER

Creates a 3D solid cylinder.

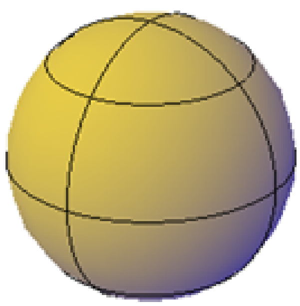
System Variables

DRAGVS

Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Create a Solid Sphere

Create a solid sphere using one of several methods.



When you start with the center point, the central axis of the sphere parallels the *Z* axis of the current user coordinate system (UCS).

Sphere Creation Options

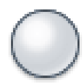
Use the following options to draw a sphere with the SPHERE command:

- **Specify three points to set the size and plane of the circumference or radius.** Use the 3P (Three Points) option to define the size of the sphere anywhere in 3D space. The three points also define the plane of the circumference.
- **Specify two points to set the circumference or radius.** Use the 2P (Two Points) option to define the size of the sphere anywhere in 3D space. The plane of the circumference matches the *Z* value of the first point.

- **Set the size and location of the sphere based on other objects.** Use the Ttr (Tangent, Tangent, Radius) option to define a sphere that is tangent to two circles, arcs, lines, and some 3D objects. The tangency points are projected onto the current UCS.

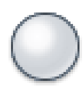
To create a solid sphere

1. Click Home tab ➤ Modeling panel ➤ Sphere.
2. Specify the center of the sphere.
3. Specify the radius or diameter of the sphere.

Toolbar: Modeling 
Command entry: SPHERE

To create a solid sphere defined by three points

1. Click Home tab ➤ Modeling panel ➤ Sphere.
2. At the Command prompt, enter **3p** (Three Points). Specify the first point.
3. Specify the second point.
4. Specify the third point.

Toolbar: Modeling 
Command entry: SPHERE

Quick Reference

Commands

SPHERE

Creates a 3D solid sphere.

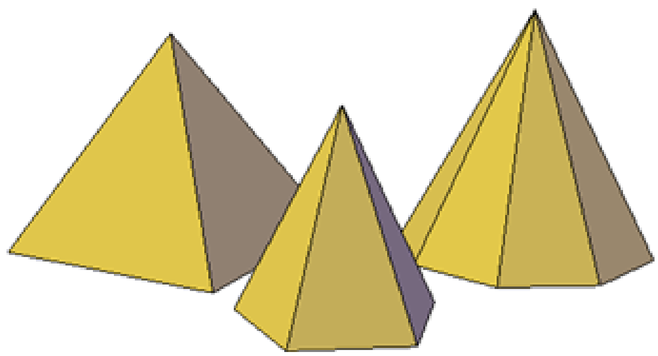
System Variables

DRAGVS

Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Create a Solid Pyramid

Create a solid pyramid with up to 32 sides.

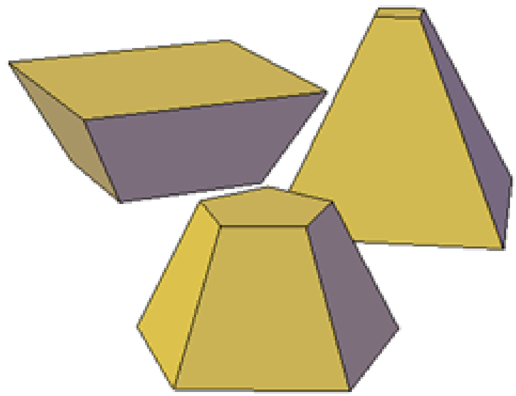


You can create a pyramid that tapers to a point, or create a frustum of a pyramid, which tapers to a planar face.

Pyramid Creation Options

Use the following options to control the size, shape, and rotation of the pyramids you create:

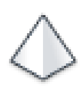
- **Set the number of sides.** Use the Sides option of the PYRAMID command to set the number of sides for the pyramid.
- **Set the length of the edges**. Use the Edges option to specify the dimension of the sides at the base.
- **Create a frustum of a pyramid.** Use the Top Radius option to create a frustum, which tapers to a planar face. The frustum face is parallel to, and has the same number of sides as, the base.



- **Set the height and rotation of the pyramid.** Use the Axis Endpoint option of the PYRAMID command to specify the height and rotation of the pyramid. This endpoint, or top of the pyramid, can be located anywhere in 3D space.

To create a solid pyramid

1. Click Home tab ➤ Modeling panel ➤ Pyramid.
2. At the Command prompt, enter **s** (Sides). Enter the number of sides to use.
3. Specify the center point of the base.
4. Specify the radius or diameter of the base.
5. Specify the height of the pyramid.

Toolbar: Modeling 
Command entry: PYRAMID

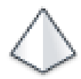
To create a frustum of a solid pyramid

1. Click Home tab ➤ Modeling panel ➤ Pyramid.
2. At the Command prompt, enter **s** (Sides). Enter the number of sides to use.
3. Specify the center point of the base.

4 Specify the radius or diameter of the base.

5 Enter **t** (Top radius). Specify the radius of the planar face at the top of the pyramid.

6 Specify the height of the pyramid.

Toolbar: Modeling 
Command entry: PYRAMID

Quick Reference

Commands

PYRAMID

Creates a 3D solid pyramid.

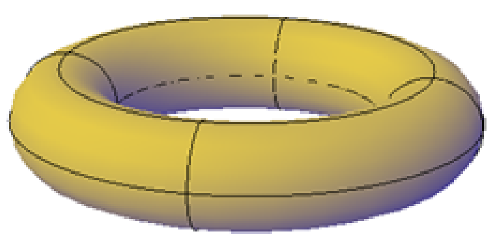
System Variables

DRAGVS

Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

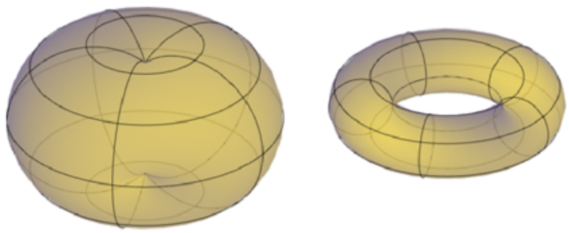
Create a Solid Torus

Create a ring-shaped solid that resembles the inner tube of a tire.



A torus has two radius values. One value defines the tube. The other value defines the distance from the center of the torus to the center of the tube. By default, a torus is drawn parallel to and is bisected by the *XY* plane of the current UCS.

A torus can be self-intersecting. A self-intersecting torus has no center hole because the radius of the tube is greater than the radius of the torus.



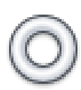
Torus Creation Options

Use the following options to control the size and rotation of the tori you create.

- **Set the size and plane of the circumference or radius.** Use the 3P (Three Points) option to define the size of the torus anywhere in 3D space. The three points also define the plane of the circumference. Use this option to rotate the torus as you create it.
- **Set the circumference or radius.** Use the 2P (Two Points) option to define the size of the torus anywhere in 3D space. The plane of the circumference matches the *Z* value of the first point.
- **Set the size and location of the torus based on other objects.** Use the Ttr (Tangent, Tangent, Radius) option to define a torus that is tangent to two circles, arcs, lines, and some 3D objects. The tangency points are projected onto the current UCS.

To create a solid torus

1. Click Home tab ➤ Modeling panel ➤ Torus.
2. Specify the center of the torus.
3. Specify the radius or diameter of the path that is swept by the torus tube.
4. Specify the radius or diameter of the tube.

Toolbar: Modeling 
Command entry: TORUS

Quick Reference

Commands

TORUS

Creates a donut-shaped 3D solid.

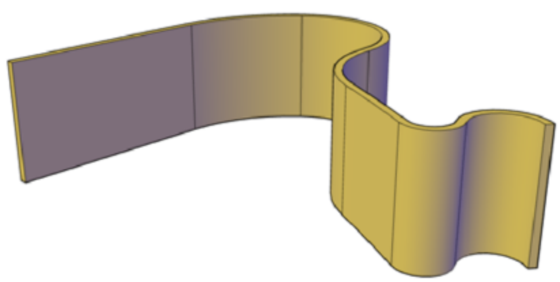
System Variables

DRAGVS

Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Create a Polysolid

Use the same techniques you use to create polylines to create a polysolid object.



The POLYSOLID command provides a quick way to draw 3D walls. A polysolid is like an extruded, wide polyline. In fact, you can draw polysolids the same way that you draw a polyline, using both straight and curved segments. Unlike extruded polylines, which lose any width properties upon extrusion, polysolids retain the width of their line segments.

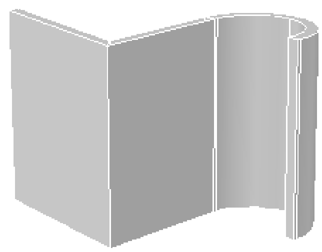
You can also convert objects such as a line, 2D polyline, arc, or circle to a polysolid.

Polysolids are displayed as swept solids in the Properties palette.

Polysolid Creation Options

Use the following options to control the size and shape of the polysolids you create:

- **Create arced segments.** Use the Arc option to add curved segments to the polysolid. The profile of a polysolid with curved segments remains perpendicular to the path.



- **Create a polysolid from a 2D object.** Use the Object option to convert an object such as a polyline, circle, line, or arc to a polysolid. The DELOBJ system variable controls whether the path (a 2D object) is automatically deleted when you create a polysolid.
- **Close the gap between the first and last points.** Use the Close option to create a connecting segment.
- **Set the height and width.** Use the Height and Width options for the POLYSOLID command. The values you set are stored in the PSOLWIDTH and PSOLHEIGHT system variables.
- **Set where the object is drawn in relation to the specified points.** Use the Justification option to place the path of the polysolid to the right, to the left, or down the center of the points you specify.

To draw a polysolid

1. Click Home tab ➤ Modeling panel ➤ Polysolid.
2. Specify a start point.
3. Specify the next point.

   To create a curved segment, at the Command prompt, enter **a** (Arc) and specify the next point.
4. Repeat step 3 to complete the desired solid.

5 Press Enter.

Toolbar: Modeling 
Command entry: POLYSOLID

To create a polysolid from an existing object

1 Click Home tab ➤ Modeling panel ➤ Polysolid.

2 At the Command prompt, enter **o** (Object).

3 Select a 2D object such as a line, polyline, arc, or circle.

   A 3D polysolid is created using the current height and width settings. The original 2D object is deleted or retained, depending on the setting of the DELOBJ system variable.

Toolbar: Modeling 
Command entry: POLYSOLID

Quick Reference

Commands

POLYSOLID

Creates a 3D wall-like polysolid.

System Variables

DELOBJ

Controls whether geometry used to create 3D objects is retained or deleted.

PSOLHEIGHT

Controls the default height for a swept solid object created with the POLYSOLID command.

PSOLWIDTH

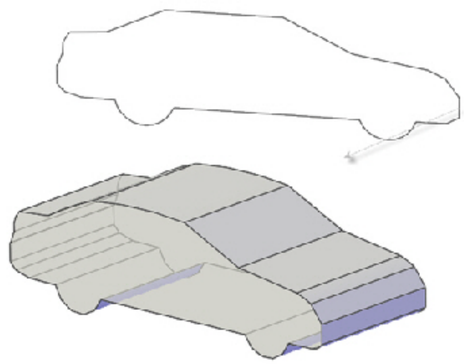
Controls the default width for a swept solid object created with the POLYSOLID command.

Construct Solids and Surfaces from Lines and Curves

Use existing lines and curves to define both the profile and path for the solid or surface.

Extrude Objects

Create solids and surfaces by extending objects into 3D space.



The EXTRUDE command creates a solid or surface that extends the shape of an object. Closed objects such as circles are converted to 3D solids. Open objects such as lines are converted to 3D surfaces.

If you extrude a polyline with width, the width is ignored and the polyline is extruded from the center of the polyline path. If you extrude an object with thickness, the thickness is ignored.

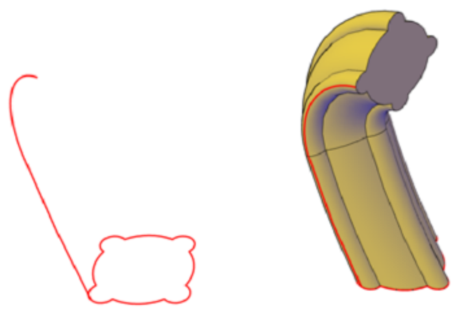
Before you can create an extruded solid from separate objects such as multiple lines or arcs, you must convert them to a single object. You can combine objects to form a polyline by using the Join option of the PEDIT command. You can also convert the objects to form a region by using the REGION command.

For information about objects that can be extruded or used as extrusion paths, see EXTRUDE.

Options for Extrusion

When you extrude objects, you can specify any of the following options:

- **Specify a path for the extrusion.** With the Path option, create a solid or surface by specifying an object to be the path for the profile, or shape, of the extrusion. The extruded solid starts from the plane of the profile and ends on a plane perpendicular to the path at the endpoint of the path. For best results, use object snaps to make sure that the path is on or within the boundary of the object being extruded.

  Extruding is different from sweeping. When you extrude a profile along a path, the profile follows the shape of the path, even if the path does not intersect the profile. With the SWEEP command, the profile moves to the location of the swept path. Sweeping usually provides greater control and better results.

- **Taper angle.** Tapering the extrusion is useful for defining parts that require a specific taper angle, such as a mold used to create metal products in a foundry.
  Avoid using large tapered angles. If the angle is too large, the profile can taper to a point before it reaches the specified height.

- **Length and direction.** With the Direction option, you can specify two points to set the length and direction of the extrusion.

To extrude an object

1. Click Home tab ➤ Modeling panel ➤ Extrude.
2. Select the objects to extrude.
3. Specify the height.

   After the extrusion, the original objects are deleted or retained, depending on the setting of the DELOBJ system variable.

Toolbar: Modeling 
Command entry: EXTRUDE

To extrude an object along a path

1. Click Home tab ➤ Modeling panel ➤ Extrude.
2. Select the objects to extrude.
3. At the Command prompt, enter **p** (Path).
4. Select the object to use as the path.

   After the extrusion, the original objects are deleted or retained, depending on the setting of the DELOBJ system variable.

Toolbar: Modeling 
Command entry: EXTRUDE

Quick Reference

Commands

EXTRUDE

Extends the dimensions of a 2D object or 3D face into 3D space.

PEDIT

Edits polylines and 3D polygon meshes.

REGION

Converts an object that encloses an area into a region object.

SWEEP

Creates a 3D solid or surface by sweeping a 2D object along a path.

System Variables

DELOBJ

Controls whether geometry used to create 3D objects is retained or deleted.

SURFU

Sets the surface density for PEDIT Smooth in the M direction and the U isolines density on surface objects.

SURFV

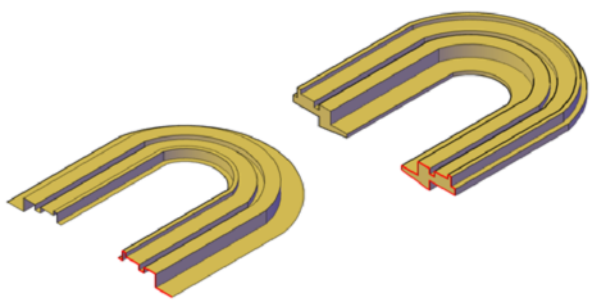
Sets the surface density for PEDIT Smooth in the N direction and the V isolines density on surface objects.

Create a Solid or Surface by Sweeping

Create a new solid or surface by sweeping a planar curve (profile) along a path.

The SWEEP command draws a solid or surface object by extending a profile shape (the swept object) along a specified path. When you sweep a profile along a path, the profile is moved and aligned normal (perpendicular) to the path.

If you sweep a closed curve along a path, the result is a solid. If you sweep an open curve along a path, the result is a surface.



You can sweep more than one profile object, but all objects must lie on the same plane.

The DELOBJ system variable controls whether the profile and sweep path are automatically deleted.

Options for Sweeping

Swept objects can be twisted or scaled as they are swept. You can also use the Properties palette to specify the following properties for the swept object:

- **Profile Rotation.** Rotates the swept profile about the path.
- **Scale Along Path.** Sets a scale factor of the end of the profile compared to the start of the profile.

- **Twist Along Path.** Sets a twist angle for the objects being swept. The value you enter sets the angle of rotation of the endpoint compared to the start point.
- **Bank (natural rotation).** Specifies whether the curve of a twisted profile rotates naturally along a 3D path.

The Properties palette does not allow changes to be made to swept properties under the following conditions:

- If the Alignment option was turned off when the profile was swept
- If the change would result in a modeling error such as a self-intersecting solid

---

TIP To sweep a profile such as a closed polyline along a helix, move or rotate the profile into place. Turn off the Alignment option in the SWEEP command. If you get a modeling error, make sure that the result will not intersect itself.

For information about what type of objects can be used to create swept solids or surfaces, see SWEEP.

See also:

- Draw Helixes on page 839
- Modify Helixes on page 1163

To create a solid or surface by sweeping an object along a path

1. Click Home tab ➤ Modeling ➤ Sweep.
2. Select objects to sweep.
3. Press Enter.
4. Select a sweep path.

   After the sweep, the original objects are deleted or retained, depending on the setting of the DELOBJ system variable.

Toolbar: Modeling 
Command entry: SWEEP

Quick Reference

Commands

SWEEP

Creates a 3D solid or surface by sweeping a 2D object along a path.

System Variables

DELOBJ

Controls whether geometry used to create 3D objects is retained or deleted.

SURFU

Sets the surface density for PEDIT Smooth in the M direction and the U isolines density on surface objects.

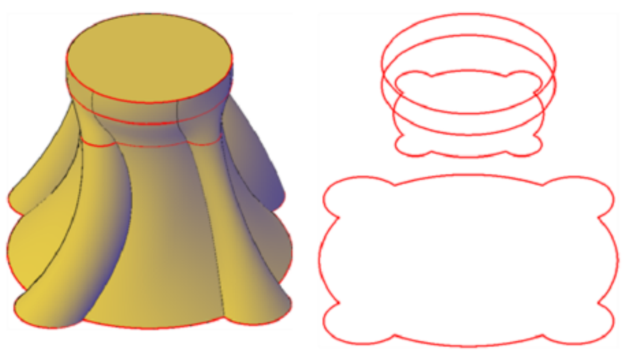
SURFV

Sets the surface density for PEDIT Smooth in the N direction and the V isolines density on surface objects.

Create a Solid or Surface by Lofting

Create a 3D solid or surface by lofting a profile through a set of two or more cross-section profiles.

The cross-section profiles define the shape of the resulting solid or surface object. You must specify at least two cross-section profiles.



Cross-section profiles can be open (for example, an arc) or closed (for example, a circle). The LOFT command flows through the space between the cross sections. If you loft through a set of a closed cross-section curves, the result is a solid object. If you loft through a set of open cross-section curves, the result is a surface object.

The cross sections that you use when lofting must be all open or all closed. You cannot use a selection set that includes both open and closed curves.

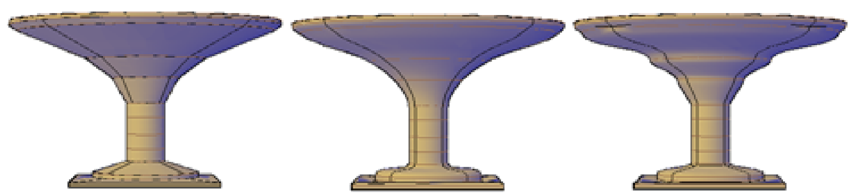
The DELOBJ system variable controls whether the cross sections, paths, and guides are automatically deleted when the solid or surface is created.

For information about objects that can be used for lofting, see LOFT.

Methods for Lofting

Lofting creates a solid or surface object that flows through other objects that define its shape.

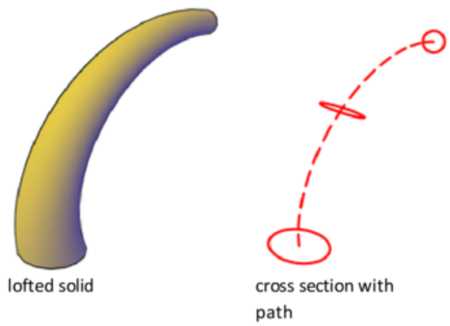
- **Cross-section profiles.** Select a series of cross-section profiles to define the shape of the new 3D object.

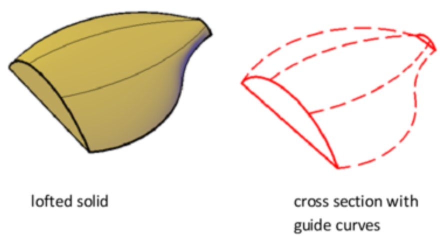
  lofted objects with different cross-section settings

  As you create a lofted object using only cross-section profiles, you can adjust its shape in the Loft Settings dialog box. You can also modify the settings later in the Properties dialog box. For more information, see Modify Properties of 3D Objects on page 1279

- **Paths.** Specify a path for the loft operation to obtain more control over the shape of the lofted object. For best results, start the path curve on the plane of the first cross section and end it on the plane of the last cross section.



- **Guide curves.** Specify guide curves to match points on corresponding cross sections. This method prevents undesired results, such as wrinkles in the resulting 3D object.

  Each guide curve must meet the following criteria:

  - 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.

To create a solid or surface by lofting through a set of cross-section profiles

1. Click Home tab ➤ Modeling panel ➤ Loft.
2. In the drawing area, select cross-section profiles and press Enter. (Select them in the order in which you want the new 3D object to pass through the cross sections.)
3. Do one of the following:
   - **Use cross-section profiles only.** Press Enter again or enter **c** (Cross sections only).

In the Loft Settings dialog box, modify the options to control the shape of the new object. Click the Preview Changes box to preview the changes as you make them.

Click OK when finished.

- **Follow guide curves.** Enter **g** (Guide curves). Select the guide curves and press Enter.
- **Follow a path.** Enter **p** (Path). Select a path and press Enter.

After the loft operation, the original objects are deleted or retained, depending on the setting of the DELOBJ system variable.

Toolbar: Modeling 
Command entry: LOFT

Quick Reference

Commands

LOFT

Creates a 3D solid or surface in the space between several cross sections.

System Variables

DELOBJ

Controls whether geometry used to create 3D objects is retained or deleted.

LOFTANG1

Sets the draft angle through the first cross section in a loft operation.

LOFTANG2

Sets the draft angle through the last cross section in a loft operation.

LOFTMAG1

Sets the magnitude of the draft angle through the first cross section in a loft operation.

LOFTMAG2

Sets the magnitude of the draft angle through the last cross section in a loft operation.

LOFTNORMALS

Controls the normals of a lofted object where it passes through cross sections.

LOFTPARAM

Controls the shape of lofted solids and surfaces.

SURFU

Sets the surface density for PEDIT Smooth in the M direction and the U isolines density on surface objects.

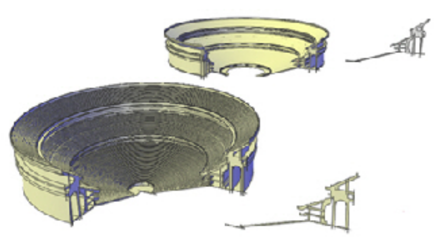
SURFV

Sets the surface density for PEDIT Smooth in the N direction and the V isolines density on surface objects.

Create a Solid or Surface by Revolving

Create a 3D object by revolving objects about an axis.

With the REVOLVE command, you can revolve open or closed objects about an axis. The revolved objects define the profile of the new solid or surface.



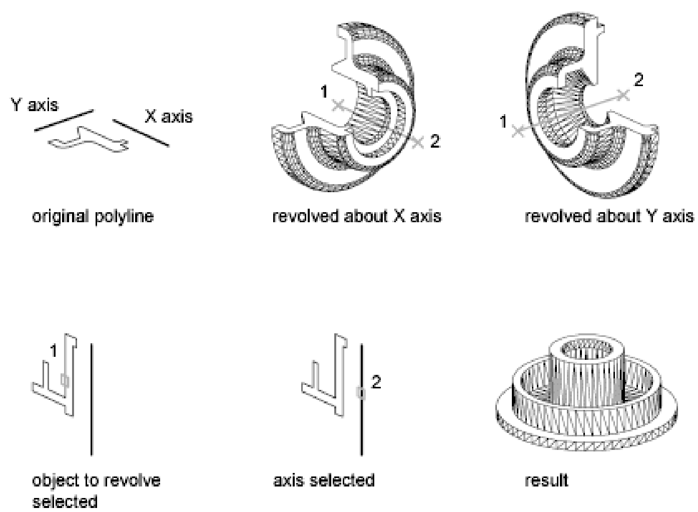
If you revolve a closed object, the result is a solid. If you revolve an open object, the result is a surface.

You can revolve more than one object at a time.

Methods for Creating a Revolved Solid or Surface

When you revolve objects, you can specify one of the following options to use as an axis of revolution:

- Axis defined by two points you specify
- *X*, *Y*, or *Z* axis
- Axis defined by an object



A profile that consists of lines or arcs that meet a polyline creates a surface object when it is revolved. To create a 3D solid object instead, first convert the profile objects to a single polyline by using the Join option of the PEDIT command.

For information about objects that can be used for revolving, see REVOLVE.

To revolve objects about an axis

1. Click Home tab ➤ Modeling panel ➤ Revolve.
2. Select the objects to revolve.
3. To specify the axis of revolution, specify one of the following:
   - **The start and endpoint.** Click to points on the screen to set the axis orientation. The axis points must be on one side of the revolved object.

The positive axis direction extends from the start point to the endpoint.

- **The X, Y, or Z axis.** Enter **x**, **y**, or **z**.
- **An object.** Select a line, a linear edge of a polyline segment, or the linear edge of a surface or solid.

4 Specify the angle of revolution.

Toolbar: Modeling 
Command entry: REVOLVE

Quick Reference

Commands

REVOLVE

Creates a 3D solid or surface by sweeping a 2D object around an axis.

System Variables

DELOBJ

Controls whether geometry used to create 3D objects is retained or deleted.

SURFU

Sets the surface density for PEDIT Smooth in the M direction and the U isolines density on surface objects.

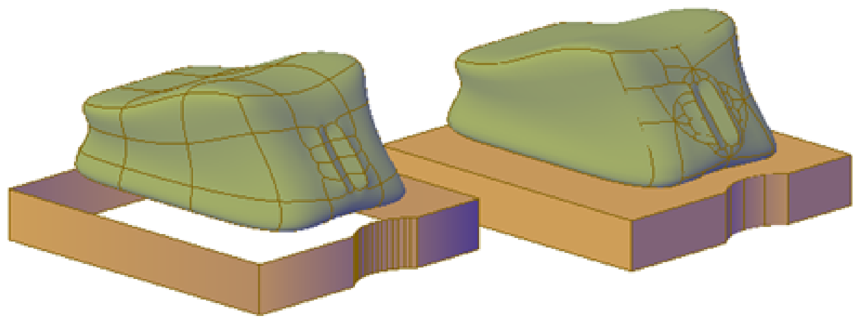
SURFV

Sets the surface density for PEDIT Smooth in the N direction and the V isolines density on surface objects.

Create 3D Solids from Objects

Convert existing objects to 3D solids.

You can use several methods to convert objects in your drawing to 3D solids.



mesh and polyline with thickness converted to optimized 3D solids

The DELOBJ system variable controls whether the objects you select are automatically deleted when the 3D object is created.

Convert Surfaces and Objects with Thickness to 3D Solids

You can convert the several types of objects into extruded 3D solids with the CONVTOSOLID command. These objects include closed polylines and circles with thickness, as well as watertight meshes and surfaces. For a complete list of objects that can be converted using this method, see CONVTOSOLID.

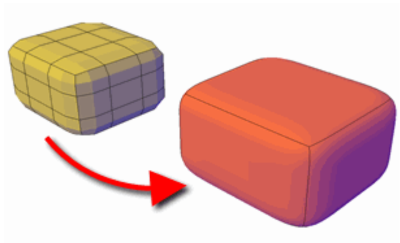
NOTE You cannot use CONVTOSOLID to convert different, contiguous objects into a 3D solid. However, you can achieve the same result by first combining them. Suppose you explode a 3D solid box into regions. Start by using CONVTOSURFACE to convert each region to a surface. Then use UNION to form a compound surface object. Finally, use CONVTOSOLID to convert the surface to a solid.

Convert Mesh to 3D Solids

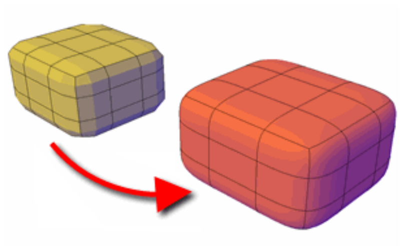
When you convert mesh objects to 3D solids, the shape of the new solid object approximates, but does not exactly duplicate, the original mesh object. You can control the differentiation somewhat by specifying whether the result is smooth or faceted (SMOOTHMESHCONVERT). You can also specify whether the resulting faces are merged (optimized).

For example, if you convert a mesh box to a solid object, you have the following options (available on the Mesh Modeling ribbon):

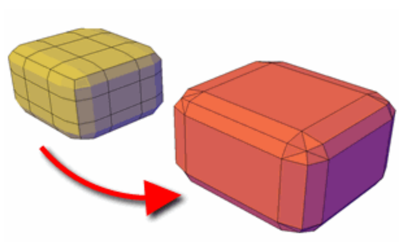
- **Smoothed and optimized.** Coplanar faces are merged into a single face. The overall shape of some faces can change. Edges of faces that are not coplanar are rounded.



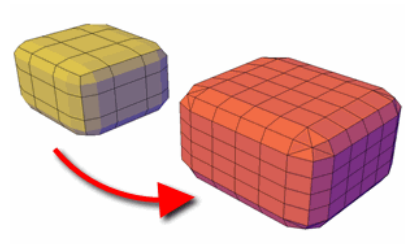
- **Smoothed and not optimized.** Each original mesh face is retained in the converted object. Edges of faces that are not coplanar are rounded.



- **Faceted and optimized.** Coplanar faces are merged into a single, flat face. The overall shape of some faces can change. Edges of faces that are not coplanar are creased, or angular.



- **Faceted and not optimized.** Each original mesh face is converted to a flat face. Edges of faces that are not coplanar are creased, or angular.



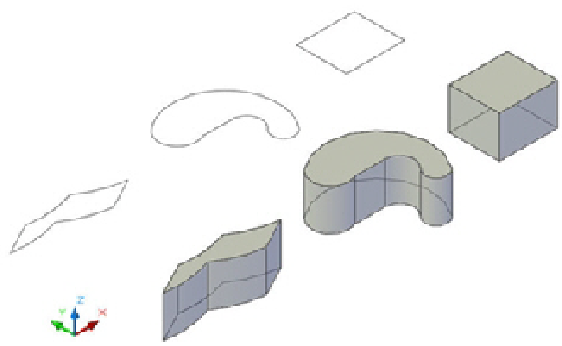
You cannot convert the following types of mesh objects to a 3D solid:

- **Mesh with gaps between faces.** Gizmo editing can sometimes result in gaps, or holes between the faces. In some cases, you can close the gaps by smoothing the mesh object.
- **Mesh that has self-intersecting boundaries.** If you have modified a mesh object so that one or more faces intersect faces in the same object, you cannot convert it to a 3D solid.

In some cases, mesh that is not eligible to be converted to a solid object can be converted to a surface.

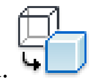
Thicken Surfaces to Convert Them to 3D Solids

You can convert 3D surface objects to 3D solids with the THICKEN command.



Grip editing is limited for objects that are created using this method.

To convert objects with thickness to extruded solids

1. Click Home tab ➤ Solid Editing panel ➤ Convert to Solid. 
2. Select one or more of the following types of objects and press Enter:
   - Uniform-width polylines with thickness
   - Closed, 0-width polylines with thickness
   - Circles with thickness

Command entry: CONVTOSOLID

To convert one or more surfaces to solids

1. Click Home tab ➤ Solid Editing panel ➤ Thicken. 
2. Select the surfaces you want to thicken. Press Enter.
3. Specify a thickness for the solid. Press Enter.

Command entry: THICKEN

To convert contiguous surfaces that enclose a volume to a 3D solid object

1. Click Home tab ➤ Solid Editing panel ➤ Union. 
2. Select objects whose surfaces are contiguous (with no gaps).
   A compound object is created.
3. Click Home tab ➤ Solid Editing panel ➤ Convert to Solid.
4. Select the new compound object.

Command entry: UNION, CONVTOSOLID

To convert a mesh object to a 3D solid

1. Click Mesh Modeling tab ➤ Convert Mesh panel ➤ Smooth with Merging.

2 Specify one of the following conversion options:
   - **Smooth, optimized.** The resulting model is smoothed, with merged faces (SMOOTHMESHCONVERT = 0).
   - **Smooth, not optimized.** The resulting model is smooth, with the same number of faces as the original mesh object (SMOOTHMESHCONVERT = 1).
   - **Faceted, optimized.** The resulting model is angular, with merged, planar faces (SMOOTHMESHCONVERT = 2).
   - **Faceted, not optimized.** The resulting model is angular, with the same number of faces as the original mesh object (SMOOTHMESHCONVERT = 3).

3 Click Mesh Modeling tab ➤ Convert Mesh panel ➤ Convert to Solid.

4 Select a mesh object that has no gaps or intersecting faces.

Command entry: SMOOTHMESHCONVERT, CONVTOSOLID

Quick Reference

Commands

CONVTOSOLID

Converts 3D meshes and polylines and circles with thickness to 3D solids.

THICKEN

Converts a surface into a 3D solid with a specified thickness.

UNION

Combines selected 3D solids, surfaces, or 2D regions by addition.

System Variables

DELOBJ

Controls whether geometry used to create 3D objects is retained or deleted.

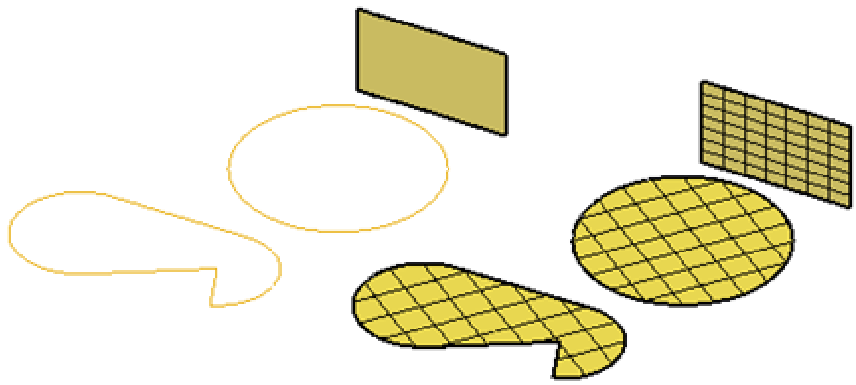
SMOOTHMESHCONVERT

Sets whether mesh objects that you convert to 3D solids or surfaces are smoothed or faceted, and whether their faces are merged.

Create Surfaces from Objects

Convert existing objects to surfaces.

You can use several methods to convert objects in your drawing to 3D surfaces.



The DELOBJ system variable controls whether the objects you select are automatically deleted when the 3D object is created.

Convert Objects to Surfaces

To convert any of the following objects into surfaces, use the CONVTOSURFACE command:

- 2D solids
- Meshes
- Regions
- Bodies
- Open, zero-width polylines with thickness
- Lines with thickness
- Arcs with thickness
- Planar 3D faces

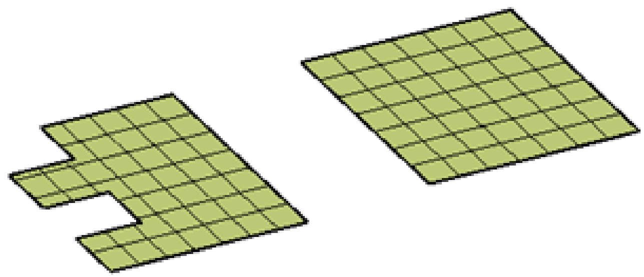
You can also create surfaces from 3D solids that have curved faces by exploding them. To convert objects such as cylinders, use the EXPLODE command.

When you convert mesh objects to surfaces, the shape of the new solid object approximates, but does not exactly duplicate, the original mesh object. You can control the differentiation somewhat by specifying whether the result is smooth or faceted (SMOOTHMESHCONVERT). You can also specify whether the resulting faces are merged (optimized).

Create a Planar Surface

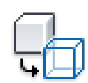
Create a flat, planar surface, use the PLANESURF command. You can use either of the following options:

- Select one or more objects that form one or more enclosed areas
- Specify the opposite corners of a rectangle



When you specify the corners of the surface, the surface is created parallel to the workplane.

To convert one or more objects to surfaces

1 Click Home tab ➤ Solid Editing panel ➤ Convert to Surface. 

2 Select the objects you want to convert and press Enter.

Command entry: CONVTOSURFACE

To create a planar surface from an existing object

1 Click Home tab ➤ Modeling panel ➤ Planar Surface.

2 At the Command prompt, enter **o** (Object).

3 Select an object and press Enter.

Toolbar: Modeling
Command entry: PLANESURF

To convert a mesh object to a surface object

1. On the Mesh Modeling tab ➤ Convert Mesh panel, specify one of the following conversion options:
   - **Smooth, optimized.** The resulting model is smoothed, with merged faces (SMOOTHMESHCONVERT = 0).
   - **Smooth, not optimized.** The resulting model is smooth, with the same number of faces as the original mesh object (SMOOTHMESHCONVERT = 1).
   - **Faceted, optimized.** The resulting model is angular, with merged, planar faces (SMOOTHMESHCONVERT = 2).
   - **Faceted, not optimized.** The resulting model is angular, with the same number of faces as the original mesh object (SMOOTHMESHCONVERT = 3).
2. Click Mesh Modeling tab ➤ Convert Mesh panel ➤ Convert to Surface.
3. Select a mesh object.

Command entry: SMOOTHMESHCONVERT, CONVTOSURFACE

To create a planar surface by specifying the corners of the surface

1. Click Home tab ➤ Modeling panel ➤ Planar Surface.
2. Specify the first corner of the surface.
3. Specify the second corner of the surface.

Toolbar: Modeling
Command entry: PLANESURF

Quick Reference

Commands

CONVTOSURFACE

Converts objects to 3D surfaces.

EXPLODE

Breaks a compound object into its component objects.

PLANESURF

Creates a planar surface.

System Variables

DELOBJ

Controls whether geometry used to create 3D objects is retained or deleted.

SMOOTHMESHCONVERT

Sets whether mesh objects that you convert to 3D solids or surfaces are smoothed or faceted, and whether their faces are merged.

Combine or Slice 3D Objects

Create new composite 3D objects or slice objects to divide them.

Create Composite Objects

Create composite 3D objects by combining, subtracting, or finding the intersecting mass of two or more 3D solids, surfaces, or regions.

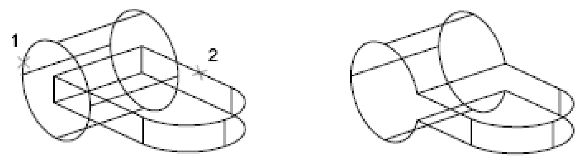
Composite solids are created from two or more solids, surfaces, or regions through any of the following commands: UNION, SUBTRACT, and INTERSECT.

By default, 3D solids record a *history* of their original forms. This history allows you to see the original forms that make up composite solids. Surface objects do not retain history. For more information, see Display Original Forms of Composite Solids on page 1326.

Methods for Creating Composite Objects

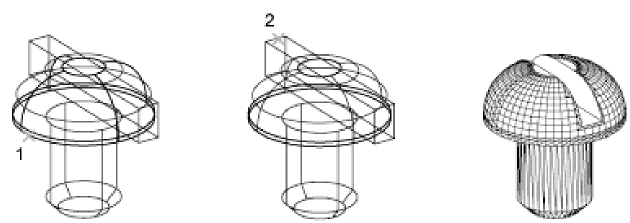
Three methods are available for creating composite solids, surfaces, or regions:

- **Combine two or more objects.**
  With UNION, you can combine the total volume of two or more objects.



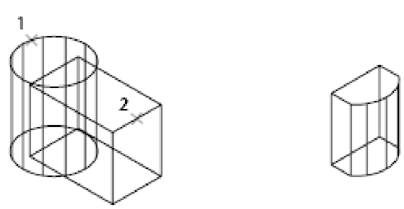
objects to be combined　　result

- **Subtract one set of solids from another.**
  With SUBTRACT, you can remove the common area of one set of solids from another. For example, you can use SUBTRACT to add holes to a mechanical part by subtracting cylinders from the object.



objects to subtract from　　objects to subtract selected　　result (lines hidden for clarity)

- **Find the common volume.**
  With INTERSECT, you can create a composite solid from the common volume of two or more overlapping solids. INTERSECT removes the portions that do not overlap and creates a composite solid from the common volume.



objects to intersect selected　　result

Create Composites from Mixed Object Types

In addition to creating composite objects from the same object types, you can also create composites from mixed surfaces and solids.

- **Mixed intersections.** Combining a solid and surface through intersection results in a surface.
- **Mixed subtractions.** Subtracting a 3D solid from a surface results in a surface. However, you cannot subtract a surface from a 3D solid object.
- **Mixed unions.** You cannot create a union between 3D solid and surface objects.

If a selection set contains objects that are both eligible and ineligible to be composite objects, the ineligible objects are ignored. For example, with the SUBTRACT command, if you select a solid object to be modified and then select both a solid and a surface to be subtracted, only the solid object is subtracted.

You cannot create composite mesh objects. However, if the selection set contains mesh objects, you can choose to convert them to 3D solids or surfaces and continue the operation. If the mesh is *watertight* (that is, it encloses a volume with no gaps), it is converted to a solid object. If the mesh has gaps, it is converted to a surface.

If a selection set of mixed objects contains regions, the regions are ignored.

To combine objects

1. Click Home tab ➤ Solid Editing panel ➤ Union.
2. Select the 3D solid, surface, or region objects to combine. Press Enter.

Toolbar: Solid Editing 
Command entry: UNION

To subtract objects from one another

1. Click Home tab ➤ Solid Editing panel ➤ Subtract.
2. Select the 3D solid, surface, or region to subtract from. Press Enter.
3. Select the 3D solid, surface, or region to subtract. Press Enter.

Toolbar: Solid Editing
Command entry: SUBTRACT

To create a compound object from the intersection with other objects

1 Click Home tab ➤ Solid Editing panel ➤ Intersect.

2 Select the 3D solid, surface, or region to intersect. Press Enter.

Toolbar: Solid Editing
Command entry: INTERSECT

Quick Reference

Commands

INTERSECT

Creates a 3D solid, surface, or 2D region from overlapping solids, surfaces, or regions.

SUBTRACT

Combines selected 3D solids, surfaces, or 2D regions by subtraction.

UNION

Combines selected 3D solids, surfaces, or 2D regions by addition.

System Variables

SHOWHIST

Controls the Show History property for solids in a drawing.

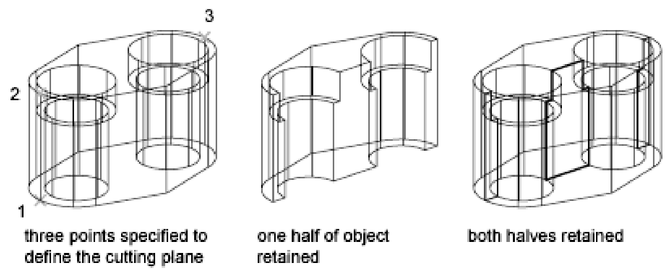
SOLIDHIST

Controls the default History property setting for new and existing objects.

Create 3D Solids or Surfaces by Slicing

Create new 3D solids or surfaces by slicing, or dividing, existing objects.

When you use the SLICE command to slice a 3D solid or surface, you can define the cutting plane in several ways. For example, you can specify three points, an axis, a surface, or a planar object to act as a cutting plane. You can retain one or both halves of the sliced object.



Sliced 3D solids do not retain a history of the original forms that created them. However, they do retain the layer and color properties of the original objects.

For a complete list of objects that can be used for a slice operation, see SLICE.

Methods for Slicing 3D Solids

You can use the following methods to define the cutting plane used to slice a 3D solid object:

- **Specify points.** The default method is to specify two points that define the cutting plane perpendicular to the current UCS. You can also specify three points.
- **Slice along a plane of the current UCS.** Specify whether you want to use the *XY*, *YZ*, or *ZX* plane.
- **Slice along the Z axis**. Specify the starting point for a slice that extends along the *Z* axis.
- **Specify a surface to act as a cutting plane.** Select a surface to act as a cutting plane. You cannot use meshes that were created using the EDGESURF, REVSURF, RULESURF, or TABSURF commands.
- **Slice along the plane of a 2D object.** Select a circle, ellipse, circular or elliptical arc, spline, or polyline segment to act as a cutting plane.

See also:

- Create Sections and 2D Drawings from 3D Models on page 1363

To slice solids or surfaces

1 Click Home tab ➤ Solid Editing panel ➤ Slice.

2 Select the 3D solid or surface objects to slice. Press Enter.

3 Specify two points to define the cutting plane.

4 Specify which side to retain, or enter **b** (Both) to retain both sides.

Command entry: SLICE

To slice solids or surfaces with a planar object

1 Click Home tab ➤ Solid Editing panel ➤ Slice.

2 Select the 3D solid or surface objects to slice. Press Enter.

3 At the Command prompt, enter **o** (Object). Press Enter.

4 Select a circle, ellipse, arc, 2D spline, or 2D polyline to use as the cutting plane.

5 Specify which side to retain, or enter **b** (Both) to retain both sides.

Command entry: SLICE

To slice solids or surfaces with a surface

1 Click Home tab ➤ Solid Editing panel ➤ Slice.

2 Select the 3D solid or surface objects to slice. Press Enter.

3 At the Command prompt, enter **s** (Surface).

4 Press Enter.

5 Select a surface to use as the cutting plane.

6 Specify which side to retain, or enter **b** (both) to retain both sides.

Command entry: SLICE

Quick Reference

Commands

SLICE

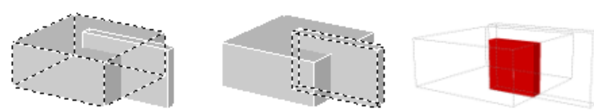
Creates new 3D solids and surfaces by slicing, or dividing, existing objects.

Check 3D Models for Interferences

Find areas where 3D solids or surfaces intersect or overlap.

Use the INTERFERE command to check for areas of interference within a set of 3D solid or surface models. You can compare two sets of objects or check all 3D solids and surfaces in a drawing.

Interference checking creates temporary solid or surface objects and highlights where the models intersect.



If the selection set contains both 3D solids and surfaces, the resulting interference object is a surface.

You cannot check interference for mesh objects. However, if you select mesh objects, you can choose to convert them to a solid or surface object and continue the operation.

During the checking operation, you can use the Interference Checking dialog box to cycle through and zoom to interference objects. You can also specify whether to delete the temporary objects that are created during interference checking.

Methods for Checking Interference

You can check interference using the following methods:

- **Define one selection set.** Check the interference of all the 3D solids and surfaces in a single selection set.
- **Define two selection sets.** Check the interference of the objects in the first set of objects against the objects in the second selection set.

- **Individually specify solids that are nested within blocks or xrefs.** Individually select 3D solid or surface objects that are nested in blocks and external references (xrefs) and compare them against other objects in the selection set.

To check for interferences within a solid model

1. Click Home tab ➤ Solid Editing panel ➤ Interference Checking.
2. Select the first set of 3D solids and surfaces in the model. Press Enter.
3. Select the second set of 3D solids and surfaces in the model. Press Enter.

   The Interference Checking dialog box is displayed. The areas of interference are displayed as new, highlighted solid objects.
4. To cycle through the interference objects, in the Interference Checking dialog box, click Next and Previous.
5. To retain the new interference objects after you close the Interference Checking dialog box, clear Delete Interference Objects Created on Close.
6. Click Close.

   If Delete Interference Objects on Close is selected, the new interference objects are deleted.

Command entry: INTERFERE

To change the display of interference objects

1. Click Home tab ➤ Solid Editing panel ➤ Interference Checking.
2. Enter **s** (Settings). Press Enter.
3. In the Interference Settings dialog box, change any of the settings and click OK.

Command entry: INTERFERE

Quick Reference

Commands

INTERFERE

Creates a temporary 3D solid from the interferences between two sets of selected 3D solids.

System Variables

INTERFERECOLOR

Sets the color for interference objects.

INTERFEREOBJVS

Sets the visual style for interference objects.

INTERFEREVPVS

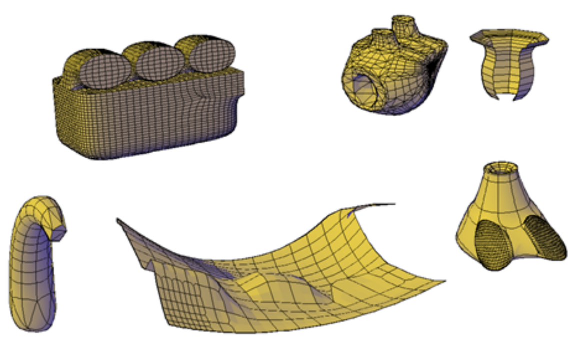
Specifies the visual style for the viewport during interference checking.

Create Meshes

Create meshes from primitive forms or by filling between points on other objects.

Overview of Creating Meshes

Mesh tessellation provides enhanced capabilities for modeling object shapes in a more detailed way.



Starting with AutoCAD 2010, the default mesh object type can be smoothed, creased, split, and refined. Although you can continue to create the legacy polyface and polygon mesh types, you can obtain more predictable results by converting to the newer mesh object type.

Methods for Creating Mesh

You can create mesh objects using the following methods:

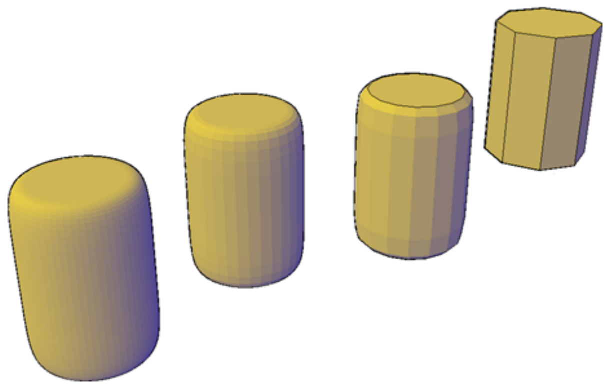
- **Create mesh primitives.** Create standard shapes, such as boxes, cones, cylinders, pyramids, spheres, wedges, and tori (MESH).
- **Create mesh from other objects.** Create ruled, tabulated, revolved, or edge-defined mesh objects, whose boundaries are interpolated from other objects or points (RULESURF, TABSURF, REVSURF, EDGESURF).
- **Convert from other object types.** Convert existing solid or surface models, including composite models, to mesh objects (MESHSMOOTH).
  You can also convert the legacy style of mesh to the new mesh object type.
- **Create custom meshes (legacy).** Use 3DMESH to create polygon meshes, usually scripted with AutoLISP routines, to create open-ended mesh. Use PFACE to create mesh with multiple vertices defined by coordinates that you specify. Although you can continue to create legacy polygonal and polyface meshes, it is recommended that you convert to the enhanced mesh object type to obtain enhanced editing capabilities.

About Tessellation

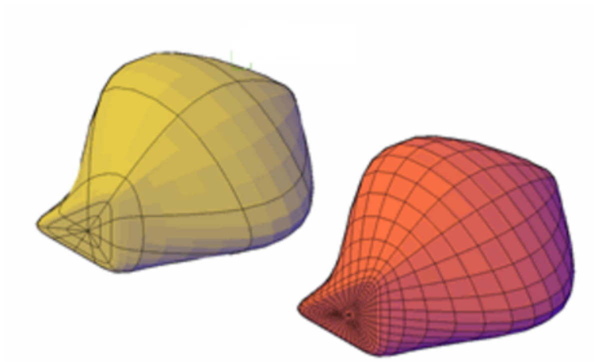
Tessellation is a collection of planar shapes that tile a mesh object. The tessellation divisions, visible in unselected mesh objects, mark the edges of the editable mesh faces. (To see these divisions in the 3D Hidden or Conceptual visual styles, VSEDGES must be set to 1.)

When you smooth and refine mesh objects, you increase the density of the tessellation (the number of subdivisions).

- **Smoothing.** Increases how closely the mesh surface adheres to a rounded form. You can increase mesh smoothness levels for selected objects in increments or by changing the smoothness level in the Properties palette. Smoothness level 0 (zero) applies the lowest level of smoothing to a mesh object. Smoothness level 4 applies a high degree of smoothness.



- **Refinement.** Quadruples the number of subdivisions in a selected mesh object or in a selected subobject, such as a face. Refinement also resets the current smoothness level to 0, so that the object can no longer be sharpened beyond that level. Because refinement greatly increases the density of a mesh, you might want to restrict this option to areas that require finely detailed modification. Refinement also helps you mold smaller sections with less effect on the overall shape of the model.



While highly refined mesh gives you the ability to make detailed modifications, it also comes at a cost: it can decrease program performance. By maintaining maximum smoothness, face, and grid levels, you can help ensure that you do not create meshes that are too dense to modify effectively. (Use SMOOTHMESHMAXLEV, SMOOTHMESHMAXFACE, and SMOOTHMESHGRID.)

Set Mesh Properties Before and After Creation

You can set defaults that control a variety of mesh properties before and after you create the mesh objects.

- Mesh Primitive Options dialog box. Sets the density of the tessellation (the number of subdivisions) per dimension for each type of mesh object you create.
- Mesh Tessellation Options dialog box. Sets the default settings for 3D solid or surface objects that you convert to mesh. Options define how closely mesh faces adhere to the shape of the object and level of smoothness. You can also set the default to prefer the settings in the Mesh Primitive Options dialog box for object conversions.
- Properties palette. Modifies properties for both the mesh object and its subobjects after they are created. For a selected mesh object, you can modify the level of smoothness. For faces and edges, you can apply or remove creasing, and modify crease retention levels.
- **Level of smoothness.** By default, the mesh primitive objects that you create have no smoothness. You can change this default with the Settings option of the MESH command. The modified smoothness value is maintained only during the current drawing session.

See also:

- Modify Mesh Objects on page 1337

To set the maximum smoothness level for mesh objects

1. At the Command prompt, enter **smoothmeshmaxlev**.
2. Enter a value from 1 to 255.

   (Use lower numbers to prevent extremely dense meshes that might affect program performance.)

To set the maximum number of faces for mesh objects

1. At the Command prompt, enter **smoothmeshmaxface**.
2. Enter a value from 1 to 16,000,000.

To control the display of the mesh facet grid

1. At the Command prompt, enter **smoothmeshgrid**.

2 Set the smoothness level at which the mesh object displays the underlying facet grid:

- **0** hides the display of the underlying facet grid.
- **1** displays the facet grid for smoothness levels 0 and 1.
- **2 or higher** specifies the highest level of smoothness at which the facet grid is displayed.

To change the default level of smoothness of new mesh primitive objects

1 At the Command prompt, enter **mesh**. Then enter **se** (SEttings).

2 Enter a smoothness value and press Enter.

(A smoothness level of 5 or less is recommended.)

3 Press Esc to end the command or specify a mesh primitive type to create.

The smoothness value is retained for the current drawing session.

Command entry: MESH

Quick Reference

Commands

3DFACE

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

3DMESH

Creates a free-form polygon mesh.

EDGESURF

Creates a mesh between four contiguous edges or curves.

MESH

Creates a 3D mesh primitive object such as a box, cone, cylinder, pyramid, sphere, wedge, or torus.

MESHOPTIONS

Displays the Mesh Tessellation Options dialog box, which controls default settings for converting existing objects to mesh objects.

MESHPRIMITIVEOPTIONS

Displays the Mesh Primitive Options dialog box, which sets the tessellation defaults for primitive mesh objects.

MESHSMOOTH

Converts 3D objects such as polygon meshes, surfaces, and solids to mesh objects.

PFACE

Creates a 3D polyface mesh vertex by vertex.

PROPERTIES

Controls properties of existing objects.

REVSURF

Creates a mesh by revolving a profile about an axis.

RULESURF

Creates a mesh that represents the surface between two lines or curves.

TABSURF

Creates a mesh from a line or curve that is swept along a straight path.

System Variables

SMOOTHMESHMAXLEV

Sets the maximum smoothness level for mesh objects.

SMOOTHMESHMAXFACE

Sets the maximum number of faces permitted for mesh objects.

SMOOTHMESHGRID

Sets the maximum level of smoothness at which the underlying mesh facet grid is displayed on 3D mesh objects.

VSEDGES

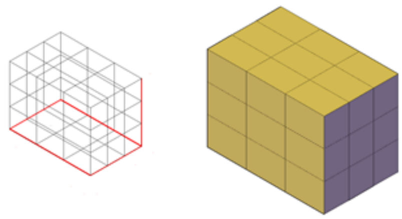
Controls the types of edges that are displayed in the viewport.

Create 3D Mesh Primitives

Create mesh boxes, cones, cylinders, pyramids, spheres, wedges, and tori.

Create a Mesh Box

Create a rectangular or cubical mesh box.



The base of the mesh box is drawn parallel to the *XY* plane of the current UCS (workplane).

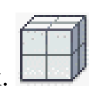
You can set defaults for the number of divisions for each dimension of new mesh boxes in the Mesh Primitive Options dialog box. You can also modify these settings and the level of smoothness as you create the mesh object.

Mesh Box Creation Options

The Box option of the MESH command provides several methods for determining the size and rotation of the mesh boxes you create.

- **Create a cube.** Use the Cube option to create a mesh box with sides of equal length.
- **Specify rotation.** Use the Cube or Length option if you want to set the rotation of the box in the *XY* plane.
- **Start from the center point.** Use the Center option to create a box using a specified center point.

To create a mesh box based on two points and a height

1. Click Mesh Modeling tab ➤ Primitives panel ➤ Mesh Box. 
2. Specify the first corner of the base.
3. Specify the opposite corner of the base.
4. Specify the height.

Toolbar: Smooth Mesh Primitives
Command entry: MESH

To create a mesh box based on length, width, and height

1 Click Mesh Modeling tab ➤ Primitives panel ➤ Mesh Box.

2 Specify the first corner of the base.

3 At the Command prompt, enter **l** (Length). Specify the length of the base.

4 Specify the width of the base.

5 Specify the height.

Toolbar: Smooth Mesh Primitives
Command entry: MESH

To create a mesh box based on a center point, corner of base, and height

1 Click Mesh Modeling tab ➤ Primitives panel ➤ Mesh Box.

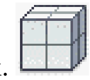
2 At the Command prompt, enter **c** (Center). Specify the center point of the base.

3 To set the location of the corner of the base, use one of the following methods:

- To set the length and width simultaneously: Specify the location of one corner of the base.
- To set the length and width separately: At the Command prompt, enter **l** (Length) and specify the length. Then specify the width.
- Specify the height.

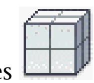
Toolbar: Smooth Mesh Primitives
Command entry: MESH

To create a mesh cube

1 Click Mesh Modeling tab ➤ Primitives panel ➤ Mesh Box. 

2 Specify the first corner or enter **c** (Center) and specify the center point of the base.

3 At the Command prompt, enter **c** (Cube). Specify the length of the cube and a rotation angle.

Toolbar: Smooth Mesh Primitives 
Command entry: MESH

Quick Reference

Commands

MESH

Creates a 3D mesh primitive object such as a box, cone, cylinder, pyramid, sphere, wedge, or torus.

MESHPRIMITIVEOPTIONS

Displays the Mesh Primitive Options dialog box, which sets the tessellation defaults for primitive mesh objects.

System Variables

DIVMESHBOXHEIGHT

Sets the number of subdivisions for the height of a mesh box along the *Z* axis.

DIVMESHBOXLENGTH

Sets the number of subdivisions for the length of a mesh box along the *X* axis.

DIVMESHBOXWIDTH

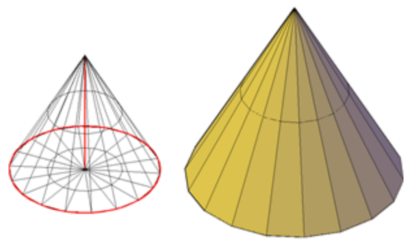
Sets the number of subdivisions for the width of a mesh box along the *Y* axis.

DRAGVS

Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Create a Mesh Cone

Create a pointed or frustum mesh cone with a circular or elliptical base.



By default, the base of the mesh cone lies on the *XY* plane of the current UCS and the height of the cone is parallel to the *Z* axis.

You can set the number of divisions for each dimension of new mesh cones in the Mesh Primitive Options dialog box. You can also modify these settings and the level of smoothness as you create the mesh object.

Mesh Cone Creation Options

The Cone option of the MESH command provides several methods for determining the size and rotation of the mesh cones you create.

- **Set the height and orientation.** Use the Axis Endpoint option when you want to reorient the cone by placing the tip or axis endpoint anywhere in 3D space.
- **Create a frustum of a cone.** Use the Top Radius option to create a frustum of a cone, which tapers to an elliptical or planar face.
- **Specify circumference and base plane.** The 3P (Three Points) option defines the size and plane of the base of the cone anywhere in 3D space.
- **Create an elliptical base.** Use the Elliptical option to create a cone base whose axes are different lengths.
- **Set the location to be tangent to two objects.** Use the Ttr (Tangent, Tangent, Radius) option to define points on two objects. Depending on the radius distance, the new cone is located as near as possible to the

tangent points you specify. You can set up tangency with circles, arcs, lines, and some 3D objects. The tangency points are projected onto the current UCS. The appearance of tangency is affected by the current level of smoothness.

To create a mesh cone with a circular base

1. Click Mesh Modeling tab ➤ Primitives panel ➤ Mesh Cone.
2. Specify the center point of the base.
3. Specify the radius or diameter of the base.
4. Specify the height of the cone.

Toolbar: Smooth Mesh Primitives
Command entry: MESH

To create a mesh cone with an elliptical base

1. Click Mesh Modeling tab ➤ Primitives panel ➤ Mesh Cone.
2. At the Command prompt, enter **e** (Elliptical).
3. Specify the start point of first axis.
4. Specify the endpoint of the first axis.
5. Specify the endpoint (length and rotation) of the second axis.
6. Specify the height of the cone.

Toolbar: Smooth Mesh Primitives
Command entry: MESH

To create a frustum of a mesh cone

1. Click Mesh Modeling tab ➤ Primitives panel ➤ Mesh Cone.

2. Specify the center point of the base.
3. Specify the radius or diameter of the base.
4. At the Command prompt, enter **t** (Top radius). Specify the top radius.
5. Specify the height of the cone.

Toolbar: Smooth Mesh Primitives
Command entry: MESH

To create a mesh cone with the height and orientation specified by the axis endpoint

1. Click Mesh Modeling tab ➤ Primitives panel ➤ Mesh Cone.
2. Specify the center point of the base.
3. Specify the radius or diameter of the base.
4. At the Command prompt, enter **a** (Axis endpoint). Specify the endpoint and rotation of the cone.

   This endpoint can be located anywhere in 3D space.

Toolbar: Smooth Mesh Primitives
Command entry: MESH

Quick Reference

Commands

MESH

Creates a 3D mesh primitive object such as a box, cone, cylinder, pyramid, sphere, wedge, or torus.

MESHPRIMITIVEOPTIONS

Displays the Mesh Primitive Options dialog box, which sets the tessellation defaults for primitive mesh objects.

System Variables

DIVMESHCONEAXIS

Sets the number of subdivisions around the perimeter of the mesh cone base.

DIVMESHCONEBASE

Sets the number of subdivisions between the perimeter and the center point of the mesh cone base.

DIVMESHCONEHEIGHT

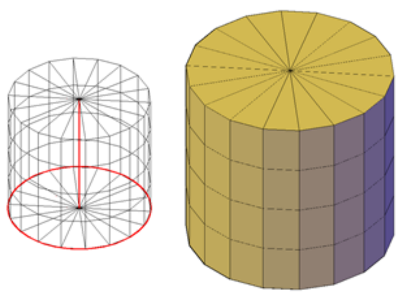
Sets the number of subdivisions between the base and the point or top of the mesh cone.

DRAGVS

Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Create a Mesh Cylinder

Create a mesh cylinder with a circular or elliptical base.



By default, the base of the mesh cylinder lies on the *XY* plane of the current UCS. The height of the cylinder is parallel to the *Z* axis.

You can set the number of divisions for each dimension of new mesh cylinders in the Mesh Primitive Options dialog box. You can also modify these settings and the level of smoothness as you create the mesh object.

Mesh Cylinder Creation Options

The Cylinder option of the MESH command provides several methods for determining the size and rotation of the mesh cylinders you create.

- **Set rotation.** Use the Axis Endpoint option to set the height and rotation of the cylinder. The center point of the top plane of the cylinder is the axis endpoint, which can be located anywhere in 3D space.
- **Use three points to define the base.** Use the 3P (Three Points) option to define the base of the cylinder. You can set three points anywhere in 3D space.
- **Create an elliptical base.** Use the Elliptical option to create a cylinder base whose axes are different lengths.
- **Set the location to be tangent to two objects.** Use the Ttr (Tangent, Tangent, Radius) option to define points on two objects. Depending on the radius distance, the new cylinder is located as near as possible to the tangent points you specify. You can set up tangency with circles, arcs, lines, and some 3D objects. The tangency points are projected onto the current UCS. The appearance of tangency is affected by the current level of smoothness.

To create a mesh cylinder with a circular base

1. Click Mesh Modeling tab ➤ Primitives panel ➤ Mesh Cylinder. 
2. Specify the center point of the base.
3. Specify the radius or diameter of the base.
4. Specify the height of the cylinder.

Toolbar: Smooth Mesh Primitives 
Command entry: MESH

To create a mesh cylinder with an elliptical base

1. Click Mesh Modeling tab ➤ Primitives panel ➤ Mesh Cylinder. 
2. At the Command prompt, enter **e** (Elliptical).

3 Specify the start point of the first axis.

4 Specify the endpoint of the first axis.

5 Specify the endpoint (length and rotation) of the second axis.

6 Specify the height of the cylinder.

Toolbar: Smooth Mesh Primitives
Command entry: MESH

To create a mesh cylinder with the height and rotation specified (axis endpoint)

1 Click Mesh Modeling tab ➤ Primitives panel ➤ Mesh Cylinder.

2 Specify the center point of the base.

3 Specify the radius or diameter of the base.

4 At the Command prompt, enter **a** (Axis endpoint). Specify the axis endpoint of the cylinder.

  This endpoint can be located anywhere in 3D space.

Toolbar: Smooth Mesh Primitives
Command entry: MESH

Quick Reference

Commands

MESH

Creates a 3D mesh primitive object such as a box, cone, cylinder, pyramid, sphere, wedge, or torus.

MESHPRIMITIVEOPTIONS

Displays the Mesh Primitive Options dialog box, which sets the tessellation defaults for primitive mesh objects.

System Variables

DIVMESHCYLAXIS

Sets the number of subdivisions around the perimeter of the mesh cylinder base.

DIVMESHCYLBASE

Sets the number of radial subdivisions from the center of the mesh cylinder base to its perimeter.

DIVMESHCYLHEIGHT

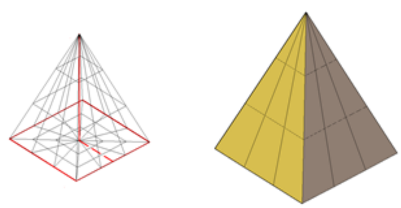
Sets the number of subdivisions between the base and the top of the mesh cylinder.

DRAGVS

Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Create a Mesh Pyramid

Create a mesh pyramid with up to 32 sides.



Create a pyramid that tapers to a point, or create a frustum of a pyramid, which tapers to a planar face.

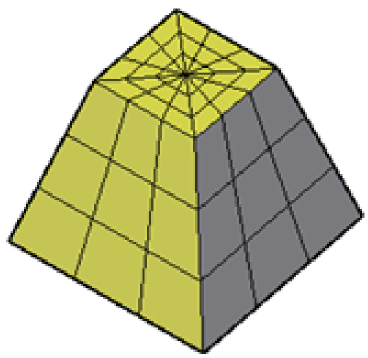
You can set the number of divisions for each dimension of new mesh pyramids in the Mesh Primitive Options dialog box. You can also modify these settings and the level of smoothness as you create the mesh object.

Mesh Pyramid Creation Options

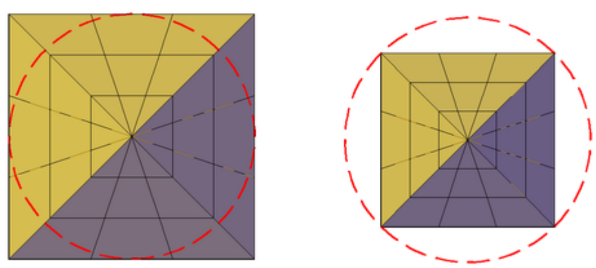
The Pyramid option of the MESH command provides several methods for determining the size and rotation of the mesh pyramids you create.

- **Set the number of sides.** Use the Sides option to set the number of sides for the mesh pyramid.

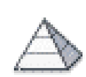
- **Set the length of the edges.** Use the Edges option to specify the dimension of the sides at the base.
- **Create a frustum of a pyramid.** Use the Top Radius option to create a frustum, which tapers to a planar face. The frustum face is parallel to, and has the same number of sides as, the base.



- **Set the height and rotation of the pyramid.** Use the Axis Endpoint option to specify the height and rotation of the pyramid. This endpoint is the top of the pyramid. The axis endpoint can be located anywhere in 3D space.
- **Set the perimeter to be inscribed or circumscribed.** Specify whether the pyramid base is drawn inside or outside of the radius.



To create a mesh pyramid

1. Click Mesh Modeling tab ➤ Primitives panel ➤ Mesh Pyramid. 
2. At the Command prompt, enter **s** (Sides). Enter the number of sides to use.

3 Specify the center point of the base.

4 Specify the radius or diameter of the base.

5 Specify the height of the pyramid.

Toolbar: Smooth Mesh Primitives 
Command entry: MESH

To create a frustum of a mesh pyramid

1 Click Mesh Modeling tab ➤ Primitives panel ➤ Mesh Pyramid. 

2 At the Command prompt, enter **s** (Sides). Enter the number of sides to use.

3 Specify the center point of the base.

4 Specify the radius or diameter of the base.

5 Enter **t** (Top radius). Specify the radius of the planar face at the top of the pyramid. Specify the height of the pyramid.

Toolbar: Smooth Mesh Primitives 
Command entry: MESH

Quick Reference

Commands

MESH

Creates a 3D mesh primitive object such as a box, cone, cylinder, pyramid, sphere, wedge, or torus.

MESHPRIMITIVEOPTIONS

Displays the Mesh Primitive Options dialog box, which sets the tessellation defaults for primitive mesh objects.

System Variables

DIVMESHPYRBASE

Sets the number of radial subdivisions between the center of the mesh pyramid base and its perimeter.

DIVMESHPYRHEIGHT

Sets the number of subdivisions between the base and the top of the mesh pyramid.

DIVMESHPYRLENGTH

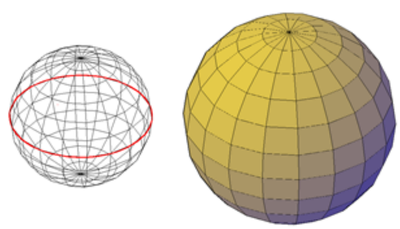
Sets the number of subdivisions along each dimension of a mesh pyramid base.

DRAGVS

Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Create a Mesh Sphere

Create a mesh sphere using one of several methods.



When you start with the center point, the central axis of the mesh sphere parallels the *Z* axis of the current user coordinate system (UCS).

You can set the number of divisions for each dimension of new mesh spheres in the Mesh Primitive Options dialog box. You can also modify these settings and the level of smoothness as you create the mesh object.

Mesh Sphere Creation Options

The Sphere option of the MESH command provides several methods for determining the size and rotation of the mesh spheres you create.

- **Specify three points to set the size and plane of the circumference or radius.** Use the 3P (Three Points) option to define the size of the sphere anywhere in 3D space. The three points also define the plane of the circumference.
- **Specify two points to set the circumference or radius.** Use the 2P (Two Points) option to define the size of the sphere anywhere in 3D space. The plane of the circumference matches the Z value of the first point.
- **Set the location to be tangent to two objects.** Use the Ttr (Tangent, Tangent, Radius) option to define points on two objects. Depending on the radius distance, the sphere is located as near as possible to the tangent points you specify. You can set up tangency with circles, arcs, lines, and some 3D objects. The tangency points are projected onto the current UCS. The appearance of tangency is affected by the current level of smoothness.

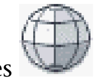
To create a mesh sphere

1. Click Mesh Modeling tab ➤ Primitives panel ➤ Mesh Sphere.
2. Specify the center of the sphere.
3. Specify the radius or diameter of the sphere.

Toolbar: Smooth Mesh Primitives
Command entry: MESH

To create a mesh sphere defined by three points

1. Click Mesh Modeling tab ➤ Primitives panel ➤ Mesh Sphere.
2. Specify the center of the sphere.
3. Specify the radius or diameter of the sphere.

Toolbar: Smooth Mesh Primitives 
Command entry: MESH

Quick Reference

Commands

MESH

Creates a 3D mesh primitive object such as a box, cone, cylinder, pyramid, sphere, wedge, or torus.

MESHPRIMITIVEOPTIONS

Displays the Mesh Primitive Options dialog box, which sets the tessellation defaults for primitive mesh objects.

System Variables

DIVMESHSPHEREAXIS

Sets the number of radial subdivisions around the axis endpoint of the mesh sphere.

DIVMESHSPHEREHEIGHT

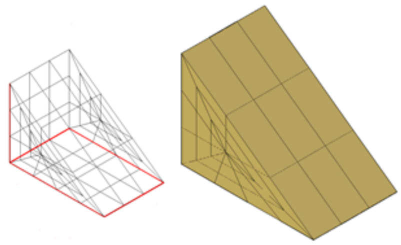
Sets the number of subdivisions between the two axis endpoints of the mesh sphere.

DRAGVS

Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Create a Mesh Wedge

Create a mesh wedge with rectangular or cubical faces.



The base of the wedge is drawn parallel to the *XY* plane of the current UCS with the sloped face opposite the first corner. The height of the wedge is parallel to the *Z* axis.

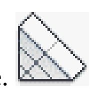
You can set the number of divisions for each dimension of new mesh wedges in the Mesh Primitive Options dialog box. You can also modify these settings and the level of smoothness as you create the mesh object.

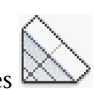
Mesh Wedge Creation Options

The Wedge option of the MESH command provides several methods for determining the size and rotation of the mesh wedges you create.

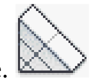
- **Create a wedge with sides of equal length.** Use the Cube option.
- **Specify rotation.** Use the Cube or Length option if you want to set the rotation of the mesh wedge in the XY plane.
- **Start from the center point.** Use the Center Point option.

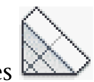
To create a mesh wedge based on two points and a height

1. Click Mesh Modeling tab ➤ Primitives panel ➤ Mesh Wedge. 
2. Specify the first corner of the base.
3. Specify the opposite corner of the base.
4. Specify the height of the wedge.

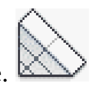
Toolbar: Smooth Mesh Primitives 
Command entry: MESH

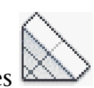
To create a mesh wedge based on length, width, and height

1 Click Mesh Modeling tab ➤ Primitives panel ➤ Mesh Wedge. 
2 Specify the first corner of the base.
3 At the Command prompt, enter **l** (Length). Specify the length of the base.
4 Specify the width of the base.
5 Specify the height of the wedge.

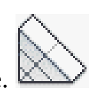
Toolbar: Smooth Mesh Primitives 
Command entry: MESH

To create a mesh wedge based on a center point, corner of base, and height

1 Click Mesh Modeling tab ➤ Primitives panel ➤ Mesh Wedge. 
2 At the Command prompt, enter **c** (Center). Specify the center point of the base.
3 To set the location of a corner of the base, use one of the following methods:
   - To set the length and width simultaneously: Specify the location of one corner of the base.
   - To set the length and width separately: At the Command prompt, enter **l** (Length) and specify the length. Then specify the width.
4 Specify the height of the wedge.

Toolbar: Smooth Mesh Primitives 
Command entry: MESH

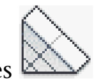
To create a mesh wedge with equal length, width, and height

1 Click Mesh Modeling tab ➤ Primitives panel ➤ Mesh Wedge. 

2 Specify the first corner or enter **c** (Center) to set the center point of the base.

3 At the Command prompt, enter **c** (Cube). Specify the length of the wedge and a rotation angle.

   The length value sets both the width and height of the wedge.

Toolbar: Smooth Mesh Primitives 
Command entry: MESH

Quick Reference

Commands

MESH

Creates a 3D mesh primitive object such as a box, cone, cylinder, pyramid, sphere, wedge, or torus.

MESHPRIMITIVEOPTIONS

Displays the Mesh Primitive Options dialog box, which sets the tessellation defaults for primitive mesh objects.

System Variables

DIVMESHWEDGEBASE

Sets the number of subdivisions between the midpoint of the perimeter of triangular dimension of the mesh wedge.

DIVMESHWEDGEHEIGHT

Sets the number of subdivisions for the height of the mesh wedge along the *Z* axis.

DIVMESHWEDGELENGTH

Sets the number of subdivisions for the length of a mesh wedge along the X axis.

DIVMESHWEDGESLOPE

Sets the number of subdivisions in the slope that extends from the apex of the wedge to the edge of the base.

DIVMESHWEDGEWIDTH

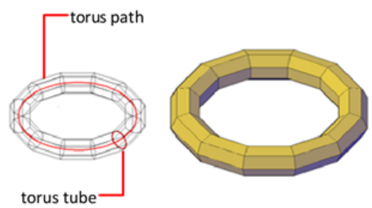
Sets the number of subdivisions for the width of the mesh wedge along the Y axis.

DRAGVS

Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Create a Mesh Torus

Create a ring-shaped solid that resembles the inner tube of a tire.



A mesh torus has two radius values. One value defines the tube. The other value defines the path, which is equivalent to the distance from the center of the torus to the center of the tube. By default, a torus is drawn parallel to and is bisected by the *XY* plane of the current UCS.

You can set the number of divisions for each dimension of new mesh tori in the Mesh Primitive Options dialog box. You can also modify these settings and the level of smoothness as you create the mesh object.

A mesh torus can be self-intersecting. A self-intersecting mesh torus has no center hole because the radius of the tube is greater than the radius of the torus.

Torus Creation Options

The Torus option of the MESH command provides several methods for determining the size and rotation of the mesh tori you create.

- **Set the size and plane of the circumference or radius**. Use the 3P (Three Points) option to define the size of the mesh torus anywhere in 3D space. The three points also define the plane of the circumference. Use this option to rotate the mesh torus as you create it.

- **Set the circumference or radius.** Use the 2P (Two Points) option to define the size of the mesh torus anywhere in 3D space. The plane of the circumference matches the *Z* value of the first point.
- **Set the location to be tangent to two objects.** Use the Ttr (Tangent, Tangent, Radius) option to define points on two objects. Depending on the specified radius distance, the path of the torus is located as near as possible to the tangent points you specify. You can set up tangency with circles, arcs, lines, and some 3D objects. The tangency points are projected onto the current UCS. The appearance of tangency is affected by the current level of smoothness.

To create a mesh torus

1. Click Mesh Modeling tab ➤ Primitives panel ➤ Mesh Torus. 
2. Specify the center of the torus.
3. Specify the radius or diameter of the overall path that is swept by the torus tube.
4. Specify the radius or diameter of the tube.

Toolbar: Smooth Mesh Primitives 
Command entry: MESH

Quick Reference

Commands

MESH

Creates a 3D mesh primitive object such as a box, cone, cylinder, pyramid, sphere, wedge, or torus.

MESHPRIMITIVEOPTIONS

Displays the Mesh Primitive Options dialog box, which sets the tessellation defaults for primitive mesh objects.

System Variables

DIVMESHTORUSPATH

Sets the number of subdivisions in the path that is swept by the profile of a mesh torus.

DIVMESHTORUSSECTION

Sets the number of subdivisions in the profile that sweeps the path of a mesh torus.

DRAGVS

Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Construct Meshes from Other Objects

Create mesh forms by filling the space between other objects such as lines and arcs.

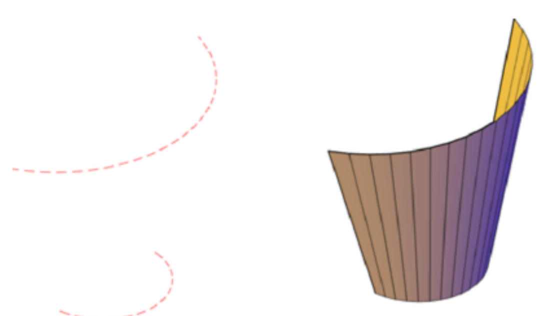
You can use a variety of methods to create mesh objects whose edges are defined by other objects. The MESHTYPE system variable controls whether the new objects are valid mesh objects, or whether they are created using legacy polyface or polygon geometry.

You can control whether the mesh is displayed as a wireframe, hidden, or conceptual image by changing the visual style (VISUALSTYLES).

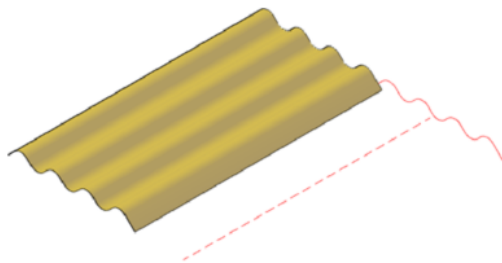
Types of Meshes Created from Other Objects

You can create several types of meshes that are based on existing objects.

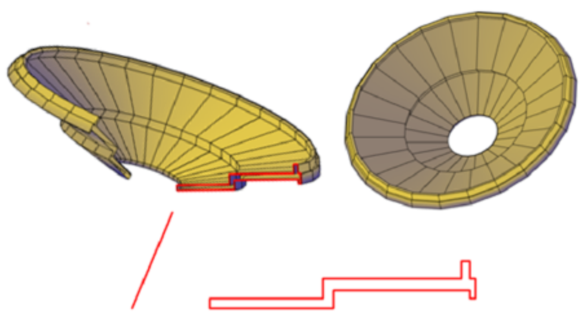
- **Ruled mesh**. RULESURF creates a mesh that represents the ruled surface between two lines or curves.



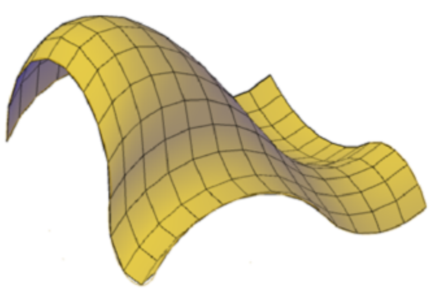
- **Tabulated mesh**. TABSURF creates a mesh that represents a general tabulated surface. The surface is defined by the extrusion of a line or curve (called a path curve) in a specified direction and distance (called a direction vector or path).



- **Revolved mesh**. REVSURF creates a mesh that approximates a surface of revolution by rotating a profile about a specified axis. A profile can consist of lines, circles, arcs, ellipses, elliptical arcs, polylines, splines, closed polylines, polygons, closed splines, and donuts.



- **Edge-defined mesh**. EDGESURF creates a mesh approximating a Coons surface patch mesh from four adjoining edges. A Coons surface patch mesh is a bicubic surface that is interpolated between four adjoining edges (which can be general space curves).

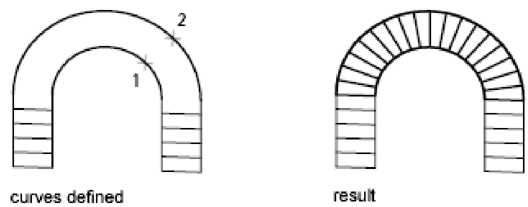


Create a Ruled Mesh

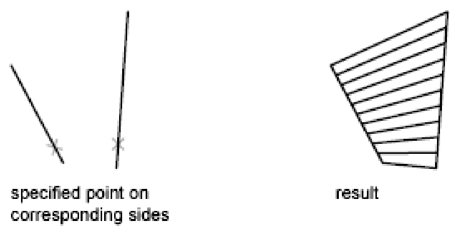
There are several methods for creating meshes.

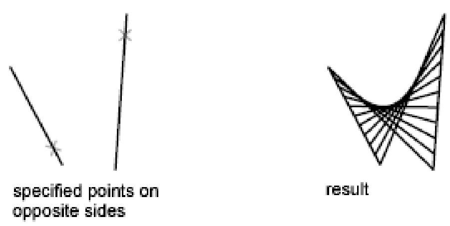
With RULESURF, you create a mesh between two lines or curves. Use two different objects to define the edges of the ruled mesh: lines, points, arcs, circles, ellipses, elliptical arcs, 2D polylines, 3D polylines, or splines.

Both objects that are used as the “rails” of a ruled mesh must be either open or closed. You can pair a point object with either an open or a closed object.



You can specify any two points on closed curves to complete the operation. For open curves, construction of the ruled mesh is based on the locations of the specified points on the curves.

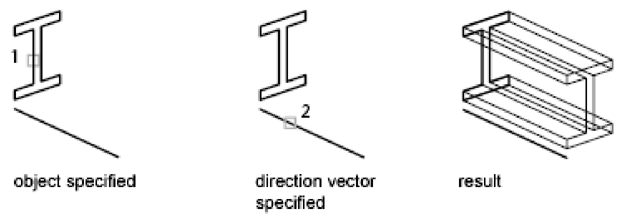




Create a Tabulated Mesh

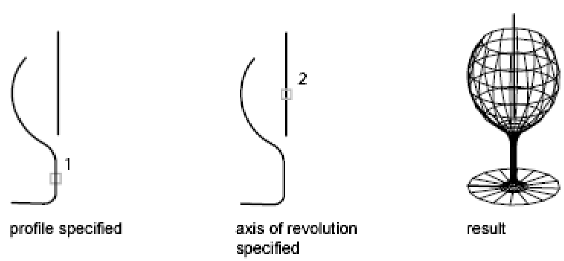
With the TABSURF command, you can create a mesh that represents a general tabulated surface defined by a path curve and a direction vector. The path curve can be a line, arc, circle, ellipse, elliptical arc, 2D polyline, 3D polyline, or spline. The direction vector can be a line or an open 2D or 3D polyline.

TABSURF creates the mesh as a series of parallel polygons running along a specified path. The original object and the direction vector must already be drawn, as shown in the following illustrations.



Create a Revolved Mesh

Use the REVSURF command to create a revolved mesh by rotating a profile of the object about an axis. REVSURF is useful for mesh forms with rotational symmetry.



The profile is called a path curve. It can consist of any combination of lines, circles, arcs, ellipses, elliptical arcs, polylines, splines, closed polylines, polygons, closed splines, or donuts.

Create an Edge-Defined Mesh

With the EDGESURF command, you can create a *Coons surface patch* mesh, as shown in the following illustration, from four objects called *edges*. Edges can be arcs, lines, polylines, splines, or elliptical arcs that form a closed loop and share endpoints. A Coons patch is a bicubic surface (one curve in the *M* direction and another in the *N* direction) interpolated between the four edges.

To create a ruled mesh

1 Click Home tab ➤ Modeling panel ➤ Ruled Surface. 

2 Select an object to act as the first defining curve.

3 Select a second object as the second defining curve.

   Mesh segments are drawn between the defining curves. The number of segments equals the value set for SURFTAB1.

4 Erase the original curves if necessary.

Command entry: RULESURF

To create a tabulated mesh

1 Click Home tab ➤ Modeling panel ➤ Tabulated Surface. 

2 Specify an object to define the overall shape of the tabulated surface (the path curve).

   The object can be a line, arc, circle, ellipse, or a 2D or 3D polyline.

3 Specify an open line or polyline that defines the direction vector.

   The mesh is extended from the start point to the endpoint of the direction vector.

4 Erase the original objects if necessary.

Command entry: TABSURF

To create a revolved mesh

1 Click Home tab ➤ Modeling panel ➤ Revolved Surface. 

2 Specify an object to define the path curve.

   The path curve, which defines the *N* direction of the mesh, can be a line, arc, circle, ellipse, elliptical arc, 2D polyline, 3D polyline, or spline. If you select a circle, closed ellipse, or closed polyline, the mesh is closed in the *N* direction.

3 Specify an object to define the axis of revolution.

The direction vector can be a line or an open 2D or 3D polyline. If you choose a polyline, the vector sets the rotation axis from its first vertex to its last vertex. Any intermediate vertices are ignored. The axis of revolution determines the *M* direction of the mesh.

4 Specify the start angle.

If you specify a nonzero start angle, the mesh is generated at a position offset from the path curve by that angle.

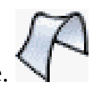
5 Specify the included angle.

The included angle specifies how far the mesh extends around the axis of revolution.

6 Erase the original objects if necessary.

Command entry: REVSURF

To create an edge-defined Coons surface patch mesh

1 Click Home tab ➤ Modeling panel ➤ Edge Surface. 

2 Select four objects to define the four adjoining edges of the mesh patch.

The objects can be arcs, lines, polylines, splines, or elliptical arcs that form a closed loop and share endpoints.

The first edge you select determines the *M* direction of the mesh.

Command entry: EDGESURF

Quick Reference

Commands

EDGESURF

Creates a mesh between four contiguous edges or curves.

PEDIT

Edits polylines and 3D polygon meshes.

PFACE

Creates a 3D polyface mesh vertex by vertex.

REVSURF

Creates a mesh by revolving a profile about an axis.

RULESURF

Creates a mesh that represents the surface between two lines or curves.

TABSURF

Creates a mesh from a line or curve that is swept along a straight path.

VISUALSTYLES

Creates and modifies visual styles and applies a visual style to a viewport.

System Variables

FACETRATIO

Controls the aspect ratio of faceting for cylindrical and conic solids.

MESHTYPE

Controls the type of mesh that is created by REVSURF, TABSURF, RULESURF and EDGESURF.

PLINECONVERTMODE

Specifies the fit method used in converting splines to polylines.

PFACEVMAX

Sets the maximum number of vertices per face.

SPLFRAME

Controls the display of splines and spline-fit polylines.

SURFTAB1

Sets the number of tabulations to be generated for the RULESURF and TABSURF commands.

SURFTAB2

Sets the mesh density in the N direction for the REVSURF and EDGESURF commands.

SURFTYPE

Controls the type of surface-fitting to be performed by the Smooth option of the PEDIT command.

SURFU

Sets the surface density for PEDIT Smooth in the M direction and the U isolines density on surface objects.

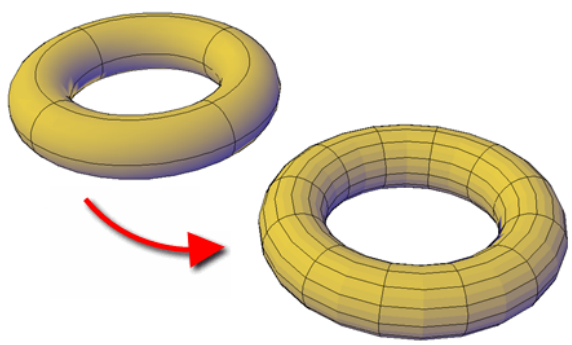
SURFV

Sets the surface density for PEDIT Smooth in the N direction and the V isolines density on surface objects.

Create Mesh by Conversion

Convert solids, surfaces, and legacy mesh types to mesh objects.

You can use the MESHSMOOTH command to convert certain objects to mesh. Convert 3D solids, surfaces, and legacy mesh objects to the enhanced mesh object in order to take advantage of capabilities such as smoothing, refinement, creasing, and splitting.



Object Types That Can Be Converted

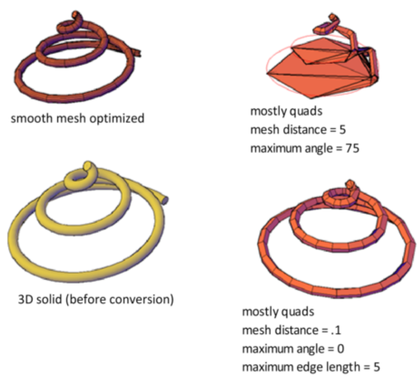
You obtain the most predictable results when you convert primitive solid objects to mesh. That is, the resulting mesh adheres closely to the shape of the original solid model.

You can also convert other types of objects, although the conversion results may differ from what you expect. These objects include swept surfaces and solids, legacy polygon and polyface mesh objects, regions, closed polylines, and objects created with 3DFACE. For these objects, you can often improve results by adjusting the conversion settings.

Adjust Mesh Conversion Settings

If the conversion does not work as expected, try changing the settings in the Mesh Tessellation Options dialog box. For example, if the Smooth Mesh Optimized mesh type results in incorrect conversions, you can set the tessellation shape to be Triangle or Mostly Quads.

You also can control the adherence to the original shape by setting the maximum distance offset, angles, aspect ratios, and edge lengths for new faces. The following example shows a 3D solid helix that has been converted to mesh using different tessellation settings. The optimized mesh version has been smoothed, but the other two conversions have no smoothness. Notice, however, that the mostly quads conversion with the lower tessellation values creates a mesh object that adheres most closely to the original version. Smoothing this object improves its appearance even more.



Similarly, if you notice that a converted mesh object has a number of long, slivered faces (which can sometimes cause gaps), try decreasing the Maximum Edge Length for New Faces value.

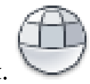
If you are converting primitive solid objects, this dialog box also offers the option of using the same default settings used to create primitive mesh objects.

When you select conversion candidates directly from this dialog box, you can preview the results before you accept them.

See also:

- Objects That Can Be Converted to Mesh

To convert objects to mesh using defaults

1 Click Home tab ➤ Mesh panel ➤ Smooth Object. 

2 Select an object such as a 3D solid or surface.

   (For a list of eligible objects, see Objects That Can Be Converted to Mesh.)

   The objects are converted to mesh using the settings in the Mesh Tessellation Options dialog box.

Toolbar: Mesh
Command entry: MESHSMOOTH

To modify conversion settings as you convert objects to mesh

1 Click Home tab ➤ Mesh panel ➤ dialog box launcher.

2 In the Mesh Tessellation Options dialog box, update the settings that you want to change.

3 Click Select Objects to Tessellate.

4 Select an object such as a 3D solid or surface and press Enter.

   (For a list of eligible objects, see Objects That Can Be Converted to Mesh.)

5 To display a preview of the converted object, click Preview.

   The updated object is displayed in the drawing area.

6 Do one of the following:

   - To adjust the settings, press Esc to display the dialog box again. Repeat steps 2 to 6.
   - To accept the conversion, press Enter.

Command entry: MESHOPTIONS

To modify mesh conversion settings

1 Click Home tab ➤ Mesh panel ➤ dialog box launcher.

2 In the Mesh Tessellation Options dialog box, update the settings that you want to change and click OK.

Command entry: MESHOPTIONS

Quick Reference

Commands

MESHOPTIONS

Displays the Mesh Tessellation Options dialog box, which controls default settings for converting existing objects to mesh objects.

MESHPRIMITIVEOPTIONS

Displays the Mesh Primitive Options dialog box, which sets the tessellation defaults for primitive mesh objects.

MESHSMOOTH

Converts 3D objects such as polygon meshes, surfaces, and solids to mesh objects.

System Variables

FACETERDEVNORMAL

Sets the maximum angle between the surface normal and contiguous mesh faces.

FACETERDEVSURFACE

Sets how closely the converted mesh object adheres to the original shape of the solid or surface.

FACETERGRIDRATIO

Sets the maximum aspect ratio for the mesh subdivisions that are created for solids and surfaces converted to mesh.

FACETERMAXEDGELENGTH

Sets the maximum length of edges for mesh objects that are created by conversion from solids and surfaces.

FACETERMAXGRID

Sets the maximum number of U and V grid lines for solids and surfaces converted to mesh.

FACETERMESHTYPE

Sets the type of mesh to be created.

FACETERMINUGRID

Sets the minimum number of U grid lines for solids and surfaces that are converted to mesh.

FACETERMINVGRID

Sets the minimum number of V grid lines for solids and surfaces that are converted to mesh.

FACETERPRIMITIVEMODE

Specifies whether smoothness settings for objects that are converted to mesh are derived from the Mesh Tessellation Options or the Mesh Primitive Options dialog box.

FACETERSMOOTHLEV

Sets the default level of smoothness for objects that are converted to mesh.

Create Custom Mesh (Legacy)

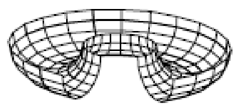
Create custom polygon or polyface mesh by specifying vertices.

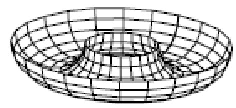
Specify individual vertices when you create mesh using the 3DMESH, PFACE, and 3DFACE commands.

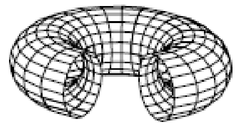
Understand Legacy Mesh Construction

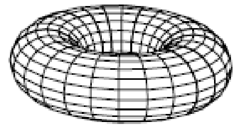
The mesh density controls the number of facets in legacy polygonal and polyface meshes. Density is defined in terms of a matrix of *M* and *N* vertices, like a grid consisting of columns and rows. *M* and *N* specify the column and row position, respectively, of any given vertex.

A mesh can be open or closed. If the start and end edges of the mesh do not touch, a mesh is open in a given direction, as shown in the following illustrations.



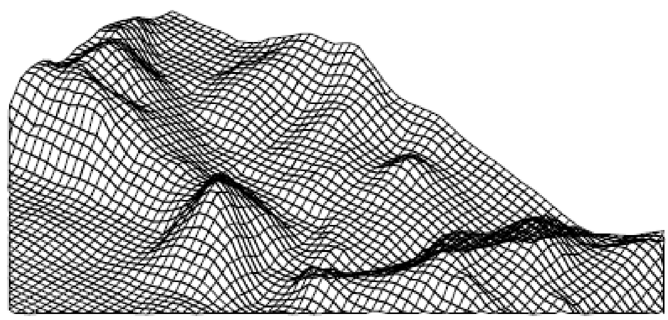




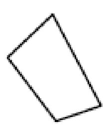


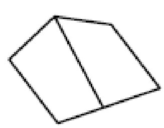
Create a Rectangular Mesh

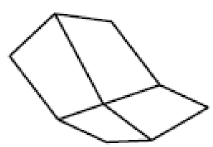
With the 3DMESH command, you can create polygon meshes that are open in both the *M* and *N* directions (like the *X* and *Y* axes of an *XY* plane). In most cases, you can use 3DMESH in conjunction with scripts or AutoLISP routines when you know the mesh points.



As you create the mesh, you specify the size of the mesh in the M and N directions. The total number of vertices you specify for the mesh is equal to the M value times the N value.





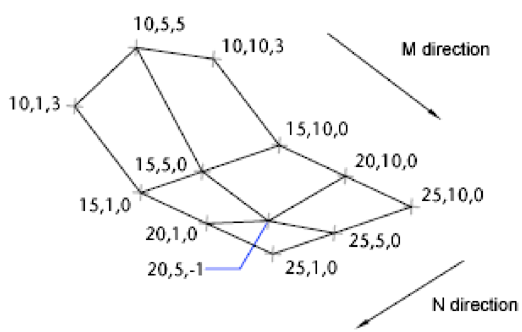


You can close the meshes with PEDIT. You can use 3DMESH to construct irregular meshes.

Example:

In the following example of text at the Command prompt, you enter the coordinate values for each vertex to create the mesh in the illustration.

Command: **3dmesh**
Mesh M size: **4**
Mesh N size: **3**
Vertex (0, 0): **10,1, 3**
Vertex (0, 1): **10, 5, 5**
Vertex (0, 2): **10,10, 3**
Vertex (1, 0): **15,1, 0**
Vertex (1, 1): **15, 5, 0**
Vertex (1, 2): **15,10, 0**
Vertex (2, 0): **20,1, 0**
Vertex (2, 1): **20, 5, -1**
Vertex (2, 2): **20,10 ,0**
Vertex (3, 0): **25,1, 0**
Vertex (3, 1): **25, 5, 0**
Vertex (3, 2): **25,10, 0**



Create a Polyface Mesh

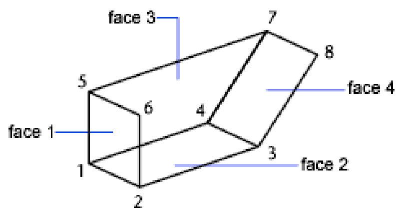
The PFACE command produces a polyface (polygon) mesh, with each face capable of having numerous vertices. PFACE is typically used by applications rather than by direct user input.

Creating a polyface mesh is like creating a rectangular mesh. To create a polyface mesh, you specify coordinates for its vertices. You then define each face by entering vertex numbers for all the vertices of that face. As you create the polyface mesh, you can set specific edges to be invisible, assign them to layers, or give them colors.

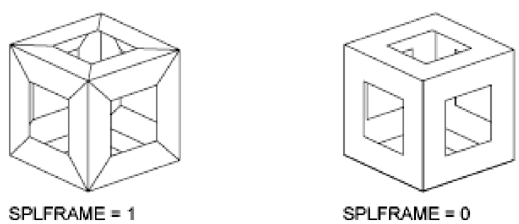
To make the edge invisible, enter the vertex number as a negative value. For instance, to make the edge between vertices 5 and 7 invisible in the following illustration, you enter the following:

Face 3, vertex 3: **-7**

In the illustration, face 1 is defined by vertices 1, 5, 6, and 2. Face 2 is defined by vertices 1, 4, 3, and 2. Face 3 is defined by vertices 1, 4, 7, and 5, and face 4 is defined by vertices 3, 4, 7, and 8.



You can control the display of invisible edges with the SPLFRAME system variable. If SPLFRAME is set to a nonzero value, the invisible edges become visible and can then be edited. If SPLFRAME is set to 0, the invisible edges remain invisible.



Create Polyface Mesh Vertex by Vertex

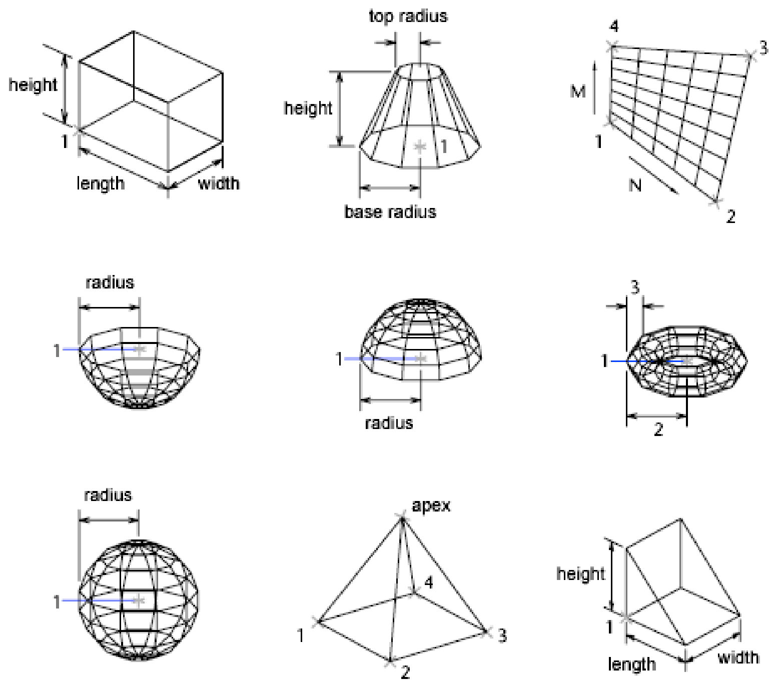
With the 3DFACE command, you can create three-dimensional polyface mesh by specifying each vertex. You can control visibility of each mesh edge segment.

If you select a 3DFACE object during some mesh smoothing operations (such as with MESHSMOOTHMORE), you are prompted to convert 3DFACE objects to mesh objects.

Create a Predefined 3D Mesh

The 3D command creates the following 3D shapes: boxes, cones, dishes, domes, meshes, pyramids, spheres, tori (donuts), and wedges.

In the following illustrations, the numbers indicate points you specify to create the mesh.



To view the objects you are creating with the 3D command more clearly, set a viewing direction with 3DORBIT, DVIEW, or VPOINT.

To create a rectangular mesh

1. Click Home tab ➤ Modeling panel ➤ 3D Mesh.
2. Specify the *M* size, using an integer from 2 through 256.
3. Specify the *N* size, using an integer from 2 through 256.
4. Specify the vertex points as prompted. The number of vertices equals the M value times the N value. Specifying the last vertex point completes the mesh.

Command entry: 3DMESH

Quick Reference

Commands

3DFACE

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

3DMESH

Creates a free-form polygon mesh.

PEDIT

Edits polylines and 3D polygon meshes.

PFACE

Creates a 3D polyface mesh vertex by vertex.

System Variables

PFACEVMAX

Sets the maximum number of vertices per face.

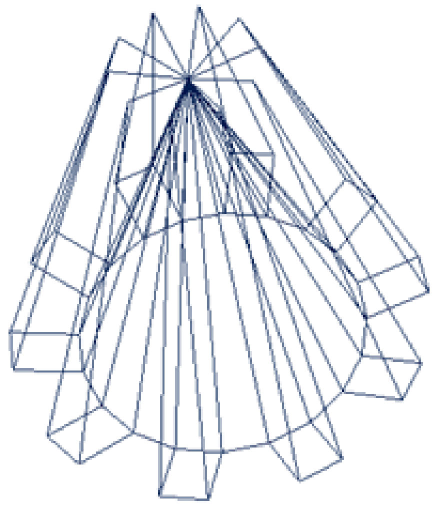
SPLFRAME

Controls the display of splines and spline-fit polylines.

Create Wireframe Models

A wireframe model is an edge or skeletal representation of a real-world 3D object using lines and curves.

You can specify a wireframe visual style to help you see the overall structure of 3D objects such as solids, surfaces, and meshes. In older drawings, you might also encounter wireframe models that were created using legacy methods.



Wireframe models consist only of points, lines, and curves that describe the edges of the object. Because each object that makes up a wireframe model must be independently drawn and positioned, this type of modeling can be the most time-consuming.

You can use a wireframe model to

- View the model from any vantage point
- Generate standard orthographic and auxiliary views automatically
- Generate exploded and perspective views easily
- Analyze spatial relationships, including the shortest distance between corners and edges, and checking for interferences
- Reduce the number of prototypes required

The ISOLINES system variable controls the number of tessellation lines used to visualize curved portions of the wireframe. The FACETRES system variable adjusts the smoothness of shaded and hidden-line objects.

Methods for Creating Wireframe Models

You can create wireframe models by positioning any 2D planar object anywhere in 3D space, using the following methods:

- Use the XEDGES command to create wireframe geometry from regions, 3D solids, surfaces, and meshes. XEDGES extracts all the edges on the selected objects or subobjects. The extracted edges form a duplicate wireframe composed of 2D objects such as lines, circles, and 3D polylines.

- Enter 3D coordinates that define the *X*, *Y*, and *Z* location of the object.
- Set the default workplane (the *XY* plane of the UCS) on which to draw the object.
- Move or copy the object to its proper 3D location after you create it.

Wireframe modeling is a skill that requires practice and experience. The best way to learn how to create wireframe models is to begin with simple models before attempting models that are more complex.

Tips for Working with Wireframe Models

Creating 3D wireframe models can be more difficult and time-consuming than creating their 2D views. Here are some tips that will help you work more effectively:

- Plan and organize your model so that you can turn off layers to reduce the visual complexity of the model. Color can help you differentiate between objects in various views.
- Create construction geometry to define the basic envelope of the model.
- Use multiple views, especially isometric views, to make visualizing the model and selecting objects easier.
- Become adept at manipulating the UCS in 3D. The *XY* plane of the current UCS operates as a workplane to orient planar objects such as circles and arcs. The UCS also determines the plane of operation for trimming and extending, offsetting, and rotating objects.
- Use object snaps and grid snap carefully to ensure the precision of your model.
- Use coordinate filters to drop perpendiculars and easily locate points in 3D based on the location of points on other objects.

To drop a perpendicular line from a 3D point down to the *XY* plane

1. Click Home tab ➤ Draw panel ➤ Line.
2. Use an object snap to specify a point on an object that is not on the *XY* plane of the UCS.

   This location defines the first point of the line.

3 Enter **.xy** and then enter **@** at the Of prompt.

   This operation extracts the *X* and *Y* coordinate values from the first point.

4 Enter **0** to specify the *Z* value.

   The *X* and *Y* values extracted from the first point are combined with a new *Z* value (0) to finish the definition of the second point.

5 Press Enter to end the command.

Toolbar: Draw
Command entry: LINE

To create wireframe geometry by extracting edges

1 Click Home tab ➤ Solid Editing panel ➤ Extract Edges.

2 Select one or more of the following objects:
   - 3D solids
   - Surfaces
   - Meshes
   - Regions
   - Edges (on 3D solids, surfaces, or meshes)
   - Faces (on 3D solids or meshes)

3 Press Enter.

   Objects such as lines, arcs, splines, or 3D polylines are created along the edges of the selected objects or subobjects.

Command entry: XEDGES

Quick Reference

Commands

UCS

Manages user coordinate systems.

XEDGES

Creates wireframe geometry from the edges of a 3D solid, surface, mesh, region, or subobject.

System Variables

ELEVATION

Stores the current elevation of new objects relative to the current UCS.

FACETRES

Adjusts the smoothness of shaded and rendered objects and objects with hidden lines removed.

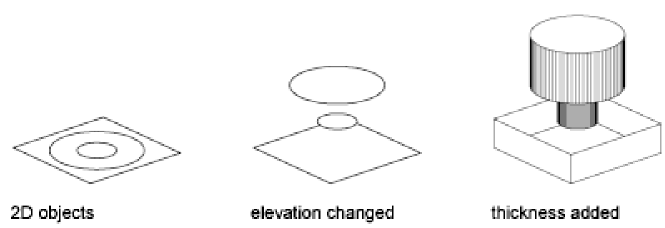
ISOLINES

Specifies the number of contour lines per surface on objects.

Add 3D Thickness to Objects

Use the thickness property to give objects a 3D appearance.

The 3D thickness of an object is the distance that object is extended, or thickened, above or below its location in space. Positive thickness extrudes upward in the positive *Z* direction; negative thickness extrudes downward (negative *Z*). Zero (0) thickness means that there is no 3D thickening of the object.



The orientation of the UCS when the object was created determines the *Z* direction. Objects with a non-zero thickness can be shaded and can hide other objects behind them.

The thickness property changes the appearance of the following types of objects:

- 2D solids
- Arcs

- Circles
- Lines
- Polylines (including spline-fit polylines, rectangles, polygons, boundaries, and donuts)
- Text (only if created as a single-line text object using an SHX font)
- Traces
- Points

Modifying the thickness property of other types of objects does not affect their appearance.

You can set the default thickness property for new objects you create by setting the THICKNESS system variable. For existing objects, change the thickness property on the Properties palette.

The 3D thickness is applied uniformly to an object; a single object cannot have different thicknesses.

You might need to change the 3D viewpoint to see the effect of thickness on an object.

NOTE Although the THICKNESS variable sets an extruded thickness for new 2D objects, those objects continue to be 2D objects. The THICKEN command adds volume to a surface object, converting it to a 3D solid.

See also:

- Create Surfaces from Objects on page 1216

To set the 3D thickness of new objects

1. Click Format menu ➤ Thickness.
2. At the Command prompt, enter the value for the thickness distance.

   When new objects are created, they have the specified 3D thickness.

Command entry: THICKNESS

To change the 3D thickness of existing objects

1. Select the objects whose 3D thickness you want to change.

2 Right-click one of the objects. Click Properties.

3 In the Properties palette, select Thickness and enter a new value.

The selected objects change to display the specified 3D thickness.

Toolbar: Standard 

Command entry: PROPERTIES

Quick Reference

Commands

ELEV

Sets elevation and extrusion thickness of new objects.

PROPERTIES

Controls properties of existing objects.

THICKEN

Converts a surface into a 3D solid with a specified thickness.

System Variables

BACKZ

Stores the back clipping plane offset from the target plane for the current viewport, in drawing units.

FRONTZ

Stores the front clipping plane offset from the target plane for the current viewport, in drawing units.

THICKNESS

Sets the current 3D thickness.

VIEWMODE

Stores the View mode for the current viewport.

Modify 3D Models

21

Change the appearance of a 3D model by manipulating both the object and its components.

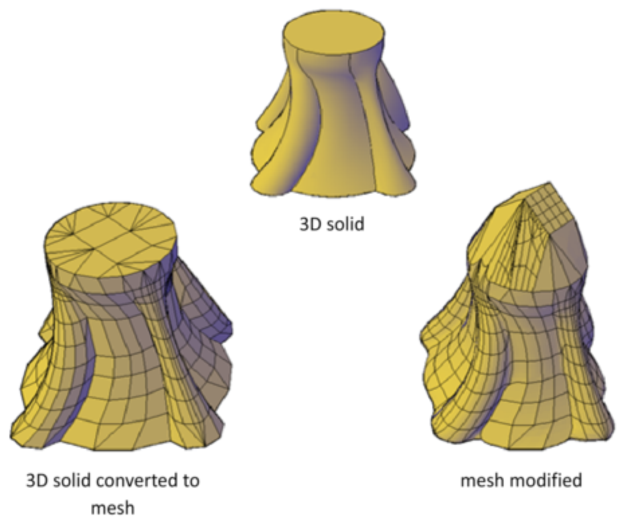
Overview of Modifying 3D Objects

Three-dimensional modeling tools range from entering precise measurements in the Properties palette, to more free-form methods such as grip and gizmo editing. Some methods are specific to 3D solids or meshes. Other methods are shared.

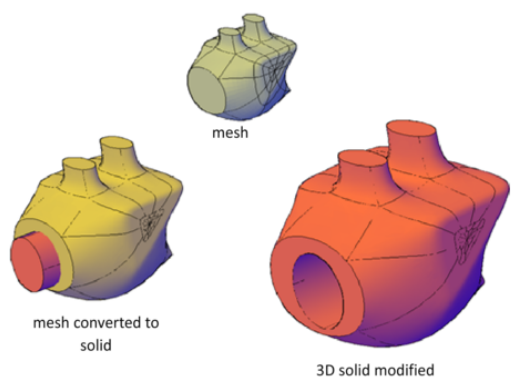
Convert to Other Object Types

In many cases, you can convert from one object type to another to take advantage of specific editing capabilities.

For example you can convert selected surfaces, solids, and legacy mesh types to mesh objects so that you can take advantage of smoothing and modeling capabilities.



Similarly, you can convert mesh to 3D solids and surfaces to accomplish some composite object modeling tasks that are available only for those objects. Conversion is often offered as a choice when you start activities that are available only for solids and surfaces.



View Your Model from All Angles

When you work with any 3D object, you can easily make changes that are not accurately reflected in the current view. To ensure that your modifications conform to your expectations, make sure you understand and use the following:

- **Manipulate the 3D workplane (UCS).** To understand how your model is projected in 3D space, learn how to use the *X*, *Y*, and *Z* axes. See Specify Workplanes in 3D (UCS) on page 650.
- **Rotate the view to display the model from different viewpoints.** Several navigation tools, including 3D Orbit and the ViewCube, are available to help you rotate around your workspace. See Use Viewing Tools on page 321.
- **Display multiple viewports.** Set up two or more viewports with different viewing angles and visual styles. When you make a change in one viewport, you can see its impact from several viewpoints at the same time. See Display Multiple Views in Model Space on page 413.

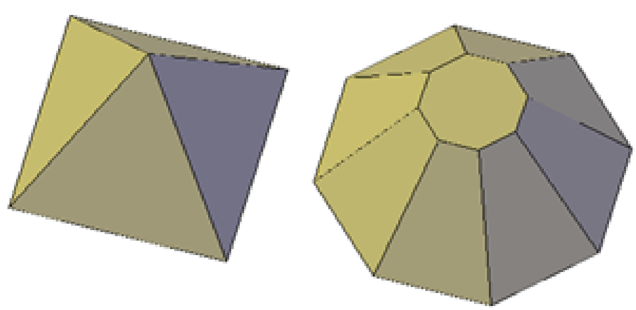
Modify Properties of 3D Objects

Modify 3D objects by changing their settings in the Properties palette.

As with other objects, you can modify the properties of 3D objects such as solids, surfaces, and mesh. In addition, you can modify specific components, known as subobjects: faces, edges, and vertices.

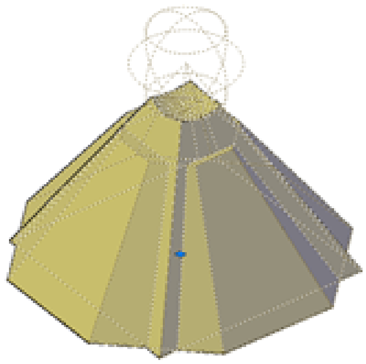
Modify the Primitive Shapes

Primitive solid forms include basic shapes such as boxes, wedges, pyramids, spheres, cylinders, cones, and tori. Each type of primitive solid has unique properties. By changing settings in the Properties palette, you can modify basic size, height, and shape characteristics. For example, to change a four-sided pyramid that ends in a point to an eight-sided pyramid that ends in a planar surface (pyramid frustum), update the Top Radius and Sides properties.



Set Whether to Retain Compound Object History

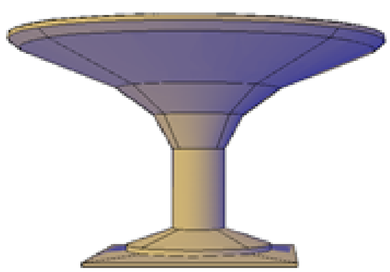
With 3D solids that have been recombined to form compound objects, you can choose to retain the history subobject, which represents components that have been removed. The Properties palette controls the availability and display of these histories. See Work with Complex 3D Solids and Surfaces on page 1326.



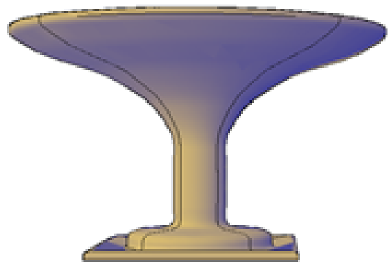
Modify Loft Settings

For lofted solids or surfaces, you can modify such properties as the number of isolines and how the profile passes through the cross sections. (These properties are set in the Loft Settings dialog box during the loft operation.) The Surface Normals property settings change the overall shape of the lofted object.

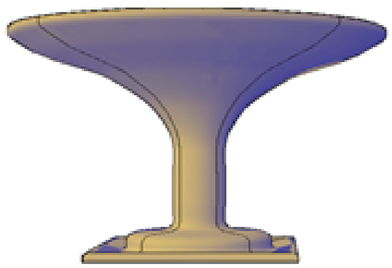
- **Ruled.** Specifies that the solid or surface is ruled between the cross sections and has sharp edges at the cross sections.



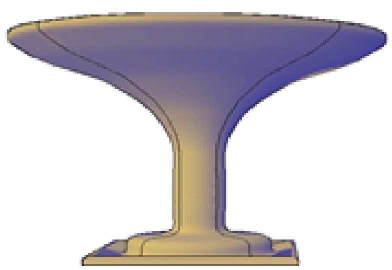
- **Smooth.** Specifies that a smooth solid or surface is interpolated between the cross sections and has sharp edges at the start and end cross sections. The adjacent cross sections control the tangent direction.



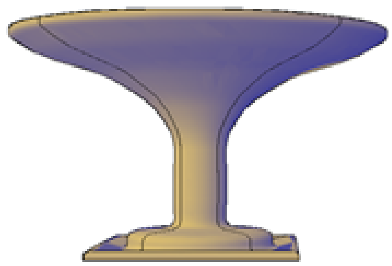
- **First Normal.** Specifies that the surface normal is perpendicular to the first cross section.



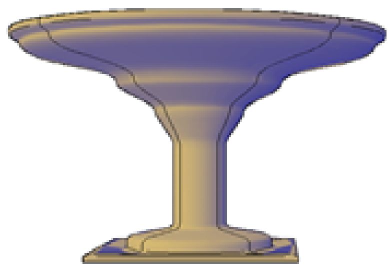
- **Last Normal.** Specifies that the surface normal is perpendicular to the last cross section.



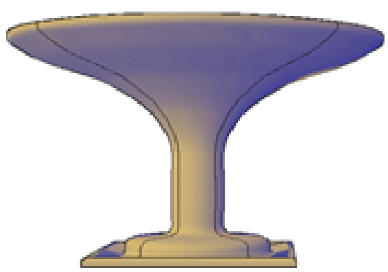
- **Ends Normal.** Specifies that the surface normal is perpendicular to both the first and last cross sections.



- **All Normal.** Specifies that the surface normal is perpendicular to all cross sections.



- **Use Draft Angles.** Controls the draft angle of the first and last cross sections of the lofted solid or surface. The following variables control the draft angle settings: LOFTANG1, LOFTANG2, LOFTMAG1, and LOFTMAG2.



Modify 3D Object and Subobject Properties

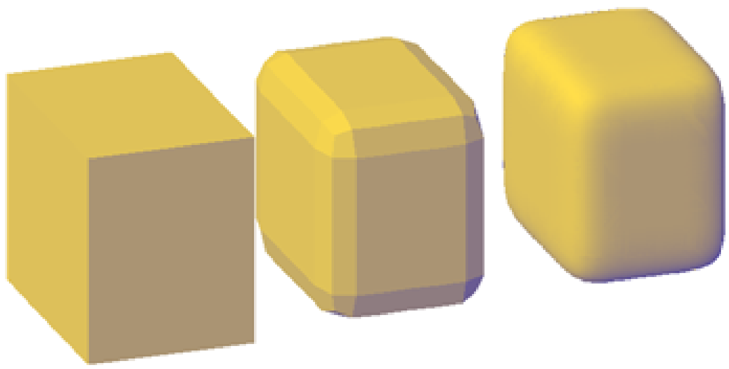
You can modify the properties of the 3D solid, surface, and mesh object in the Properties palette. In addition, you can modify specific properties of individual subobjects, such as faces, edges, and vertices. Different properties are available for different types of subobjects.

In some cases, the application of properties can differ. depending on the object type. For example, you can modify the properties of mesh faces, including their color. However, the color appearance of a mesh face might differ from the equivalent color on a 3D solid face. This difference occurs because changing the color of a face modifies the diffuse color of the face, but not the ambient color (which is derived from the mesh material property). To obtain a closer match between the color of 3D solid and mesh faces, you can add lights and turn off the default lighting (which disables ambient lighting). You can also try assigning a material that has the same ambient and diffuse color. For more information see Create Materials.

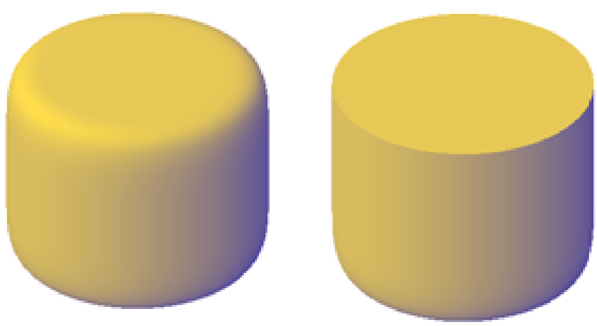
Modify Mesh Forms by Changing Properties

Mesh objects have additional properties that control the level of smoothness and creases. Crease properties of face, edge, and vertex subobjects are also reflected in the Properties palette.

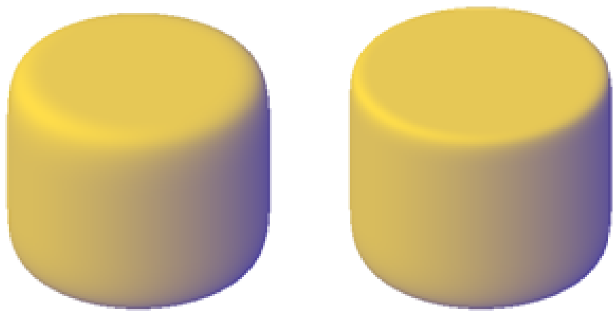
- **Level of Smoothness.** Smooths or sharpens the edges of a mesh object.



- **Crease Type.** Specifies the presence of a crease (or sharpened edge) and the effect of smoothing. Smoothing does not affect a crease with a value of Always. A crease set to By Level retains its sharpness until the mesh object is smoothed to the specified crease level.



- **Crease Level.** When a crease is set to By Level, indicates the smoothing level at which the crease starts to lose its sharpness.



To modify the contour of a lofted solid or surface by changing the surface normal settings (Properties palette)

1. In a drawing, select a lofted solid or surface that was created with cross sections.
2. If the Properties palette is not displayed, select any object. Right-click the object to display the shortcut menu. Click Properties.
3. On the Properties palette, Geometry area, change the Surface Normals setting. (For a description, see Modify Loft Settings on page 1280.)

To modify a mesh object in the Properties palette

1. If the Properties palette is not displayed, select any object. Right-click the object to display the shortcut menu. Click Properties.
2. Click the mesh object to select it.
3. In the Properties palette, modify the properties you want to change.

Command entry: PROPERTIES

To modify a mesh face, edge, or vertex in the Properties palette

1. If the Properties palette is not displayed, select any object. Right-click the object to display the shortcut menu. Click Properties.
2. Press Ctrl+click the mesh face, edge, or vertex that you want to modify.

NOTE If you cannot select a specific subobject, verify whether subobject selection filtering is turned on for a different subobject type. (Right-click in the drawing area and click Subobject Selection Filter.)

3 In the Properties palette, modify the properties you want to change.

Command entry: PROPERTIES

Quick Reference

Commands

PROPERTIES

Controls properties of existing objects.

System Variables

LOFTANG1

Sets the draft angle through the first cross section in a loft operation.

LOFTANG2

Sets the draft angle through the last cross section in a loft operation.

LOFTMAG1

Sets the magnitude of the draft angle through the first cross section in a loft operation.

LOFTMAG2

Sets the magnitude of the draft angle through the last cross section in a loft operation.

Use Grips and Gizmos to Modify 3D Models

Use grips and gizmos to change the shape and size of solids and surfaces.

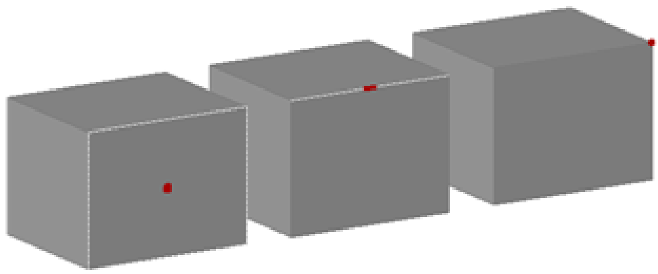
Select 3D Subobjects

Select faces, edges, and vertices by pressing Ctrl as you select 3D objects.

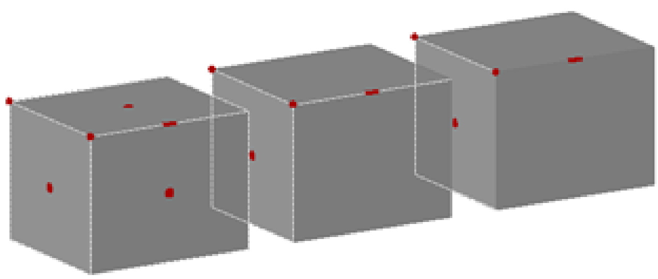
A *subobject* is a face, edge or vertex of a solid, surface, or mesh object.

Select Subobjects

To select a face, edge, or vertex of a 3D object, press Ctrl as you select the object. Selected subobjects display different types of grips, depending on the subobject type.



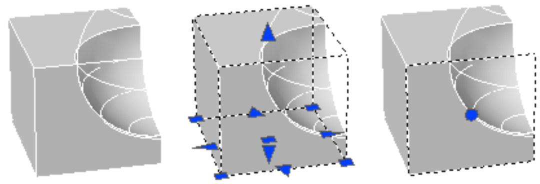
You can select one or more subobjects on any number of 3D objects. The selection set can include more than one type of subobject. Press Ctrl to select subobjects at the selection prompts of the MOVE, ROTATE, SCALE, and ERASE commands.



You can remove an item from the selection set by pressing the Shift key and selecting it again.

Select Subobjects on Composite 3D Solids

Press and hold Ctrl to select faces, edges, and vertices on composite solids. If the History property of the composite solid is set to Record, the first "pick" might select the *history subobject*. (The history subobject is the portion of the original object that was removed during the union, subtract, or intersect operation.) Continue to hold down Ctrl and pick again to select a face, edge, or vertex on the original form.

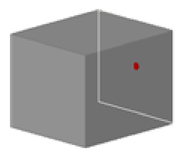


Cycle Through Multiple Subobjects

Selection sets can contain any number of 3D solids, surfaces, meshes, and subobjects.

In 3D views, some objects or subobjects might be hidden behind others. You can press Ctrl+Spacebar to cycle through the hidden subobjects until the object you want to select is highlighted.

For example, when you select faces on a box, the face in the foreground is detected first. To select a hidden face, press the Spacebar (with Ctrl still pressed). Release the Spacebar and click to select the face.



For best results, make sure that Selection Preview is turned on in the Options dialog box. If selection preview is turned off, you can cycle through hidden subobjects by pressing Ctrl+Spacebar as you click until the subobject is selected.

Turn on the Subobject Selection Filter

Selecting a specific type of subobject can be difficult on complex objects, such as meshes. You can limit the selection to a face, edge, vertex, or history subobject by setting a subobject selection filter.

Stored in the SUBOBJSELECTIONMODE system variable, this setting can be changed from the shortcut menu or on the ribbon.

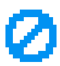
When a subobject filter is turned on, the following icons are displayed near the cursor:


Vertex filtering is on


Edge filtering is on


Face filtering is on


History subobject filtering is on


Subobject not eligible for selection

To select one or more faces, edges, or vertices on a 3D object

1. Press Ctrl+click a face, edge, or vertex.
2. Repeat step 1 until all subobjects are selected.

To create a selection set of 3D objects and subobjects

1. Click the required solid and surface objects.
2. Press and hold Ctrl.
3. Click a face, edge, or vertex.
4. Repeat steps 1 through 3 until the required subobjects are selected.

Command entry: SELECT

To remove a subobject from a selection set

1. Press and hold Ctrl+Shift.
2. Click a selected face, edge, or vertex.
   The grip, if displayed, changes from red to blue.

To select a face, edge, or vertex on the history form of a composite solid

1. Press and hold Ctrl.
2. Select the original form on which you want to select a face, edge, or vertex.
3. With Ctrl still pressed, click a face, edge, or vertex.

   You might need to repeat this step to select the required face, edge, or vertex.

Command entry: SELECT (Subobject option)

To cycle through and select subobjects that are overlapping

1. Press and hold Ctrl as you press the Spacebar to cycle through the subobjects that are both visible and hidden.
2. If the subobject that you want to select is not highlighted during the cycle, move the cursor and repeat.
3. When the subobject is highlighted, release the Spacebar and click.

   The subobject is selected.

To limit selection to a specific type of subobject

1. Click Home tab ➤ Subobject panel ➤ No Filter, Edge, Vertex, Face, or Solid History.
2. Press Ctrl+click a 3D solid or mesh subobject.

   Only the subobject type that you specified can be selected. If you specified None, all subobject types can be selected.

Shortcut menu: Subobject Selection Filter
Command entry: SUBOBJSELECTIONMODE

Quick Reference

Commands

ERASE

Removes objects from a drawing.

MOVE

Moves objects a specified distance in a specified direction.

ROTATE

Rotates objects around a base point.

SCALE

Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling.

SELECT

Places selected objects in the Previous selection set.

SOLIDEDIT

Edits faces and edges of 3D solid objects.

System Variables

GRIPHOVER

Controls the fill color of an unselected grip when the cursor pauses over it.

GRIPOBJLIMIT

Suppresses the display of grips when the selection set includes more than the specified number of objects.

GRIPS

Controls the use of selection set grips for the Stretch, Move, Rotate, Scale, and Mirror Grip modes.

GRIPSIZE

Sets the size of the grip box in pixels.

GRIPSUBOBJMODE

Sets whether edge, face, or vertex grips are active upon initial selection.

LEGACYCTRLPICK

Specifies the keys for selection cycling and the behavior for Ctrl+click.

SUBOBJSELECTIONMODE

Filters whether a face, edge, or vertex is selected with Ctrl+click.

Use Grips to Edit 3D Solids and Surfaces

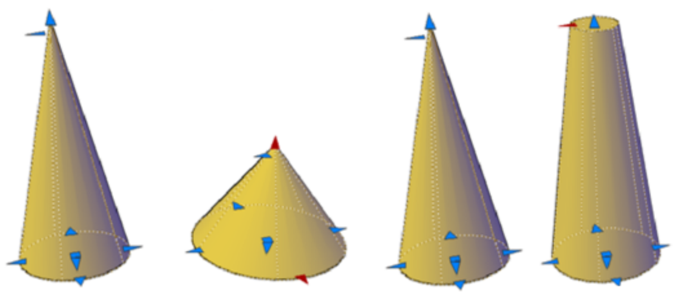
Use grips to change the size and shape of some individual solids and surfaces.

The method you use to manipulate the 3D solid or surface depends on the type of object and the method used to create it.

NOTE For mesh objects, only the center grip is displayed. However you can edit mesh objects with the 3D Move, Rotate, or Scale gizmos.

Primitive Solid Forms and Polysolids

You can drag grips to change the shape and size of primitive solids and polysolids. For example, you can change the height and base radius of a cone without losing the overall cone shape. Drag the top radius grips to transform the cone to a flat-topped, frustum cone.



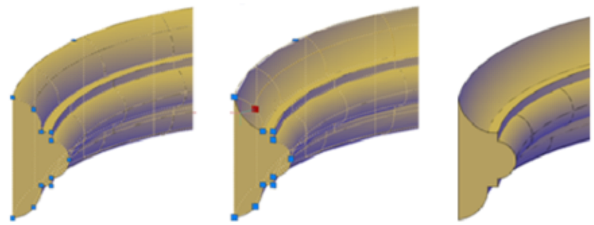
Extruded Solids and Surfaces

You can convert 2D objects to solids and surfaces with the EXTRUDE command. When selected, extruded solids and surfaces display grips on their profiles. A *profile* is the original outline that defines the shape of the extruded solid or surface. Drag profile grips to modify the overall shape of the object.

If the extrusion was created along a sweep path, the path can be manipulated with grips. If a path was not used, you can modify the height of the object using a grip at the top of the extruded solid or surface.

Swept Solids and Surfaces

Swept solids and surfaces display grips on the swept profile as well as on the sweep path. You can drag these grips to modify the solid or surface.



swept polyline | polyline vertex stretched to change profile | swept polyline with modified profile

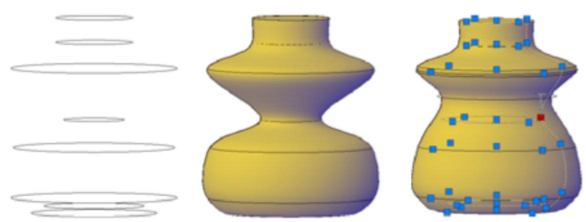
When you click and drag a grip on the profile, the changes are constrained to the plane of the profile curve.

Lofted Solids and Surfaces

Depending on how a lofted solid or surface was created, the solid or surface displays grips on the following, defining lines or curves:

- Cross section
- Path

Drag grips on any of the defining lines or curves to modify the shape. If the lofted object contains a path, you can only edit the portion of the path that is between the first and last cross sections.



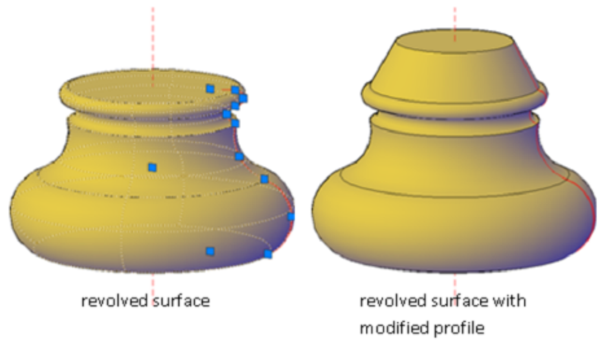
cross sections | lofted solid | lofted solid with modified cross section

You cannot use grips to modify lofted solids or surfaces that are created with guide curves.

Revolved Solids and Surfaces

Revolved solids and surfaces display grips on the revolved profile at the start of the revolved solid or surface. You can use these grips to modify the profile of the solid of surface.

A grip is also displayed at the axis of revolution endpoint. You can relocate the axis of revolution by dragging the grip to another location.



See also:

- Create 3D Solids and Surfaces on page 1175
- Create a Solid Cone on page 1185

To relocate the axis of revolution for a revolved solid or surface

1. In a drawing, select a revolved solid or surface.
2. Select the grip on the axis of revolution.
3. Click in another location to relocate the axis of revolution.

Quick Reference

Commands

EXTRUDE

Extends the dimensions of a 2D object or 3D face into 3D space.

SELECT

Places selected objects in the Previous selection set.

System Variables

GRIPHOVER

Controls the fill color of an unselected grip when the cursor pauses over it.

GRIPOBJLIMIT

Suppresses the display of grips when the selection set includes more than the specified number of objects.

GRIPS

Controls the use of selection set grips for the Stretch, Move, Rotate, Scale, and Mirror Grip modes.

GRIPSIZE

Sets the size of the grip box in pixels.

GRIPSUBOBJMODE

Sets whether edge, face, or vertex grips are active upon initial selection.

Use Gizmos to Modify Objects

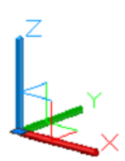
Use gizmos to move, rotate, or scale objects and subobjects in a 3D view.

Overview of Using Gizmos

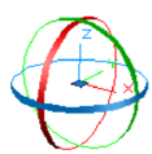
Gizmos help you move, rotate, or scale a set of objects along a 3D axis or plane.

There are three types of gizmos:

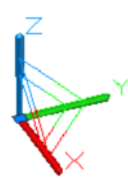
- **3D Move gizmo.** Relocates selected objects along an axis or plane.
- **3D Rotate gizmo.** Rotates selected objects about a specified axis.
- **3D Scale gizmo.** Scales selected objects along a specified plane or axis, or uniformly along all 3 axes.



3D Move gizmo



3D Rotate gizmo



3D Scale gizmo

By default, gizmos are displayed automatically when you select an object or subobject in a view that has a 3D visual style. Because they constrain modifications along specific planes or axes, gizmos help ensure more predictable results.

You can specify which gizmos are displayed when an object is selected, or you can suppress their display.

Quick Reference

Commands

3DMOVE

In a 3D view, displays the 3D Move gizmo to aid in moving 3D objects a specified distance in a specified direction.

3DROTATE

In a 3D view, displays the 3D Rotate gizmo to aid in revolving 3D objects around a base point.

3DSCALE

In a 3D view, displays the 3D Scale gizmo to aid in resizing 3D objects.

System Variables

DEFAULTGIZMO

Sets the 3D Move, 3D Rotate, or 3D Scale gizmo as the default during subobject selection.

GRIPSUBOBJMODE

Sets whether edge, face, or vertex grips are active upon initial selection.

GTAUTO

Controls whether 3D gizmos are automatically displayed when you select objects before you start a command in a viewport with a 3D visual style.

GTDEFAULT

Controls whether the 3D Move, 3D Rotate, or 3D Scale operation starts automatically when you start the MOVE, ROTATE, or SCALE command in a viewport with a 3D visual style.

GTLOCATION

Controls the initial location of the 3D Move, 3D Rotate, or 3D Scale gizmo when you select objects before you start a command in a viewport with a 3D visual style.

Use the Gizmos

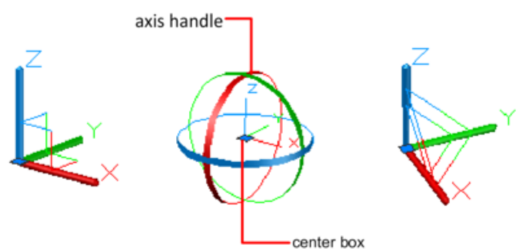
Gizmos help move, rotate, and scale 3D objects and subobjects.

Display the Gizmos

Gizmos are available only in 3D views that are set to use a 3D visual style such as 3D Hidden. You can set the gizmo to be displayed automatically when you select a 3D object or subobject. Gizmos are also displayed during the 3DMOVE, 3DROTATE, and 3DSCALE commands.

If the visual style is set to 2D Wireframe, entering 3DMOVE, 3DROTATE, or 3DSCALE automatically converts the visual style to 3D Wireframe.

By default, the gizmo is initially placed in the center of the selection set. However, you can relocate it anywhere in 3D space. The center box (or base grip) of the gizmo sets the base point for the modification. This behavior is equivalent to temporarily changing the position of the UCS as you move or rotate the selected objects. The axis handles on the gizmo constrain the movement or rotation to an axis or plane.



For best results, use object snaps to locate the grip center box.

Switch Between the Gizmos

Whenever you select an object in a 3D view, the default gizmo is displayed. You can select a different default on the ribbon, or change the value of the the DEFAULTGIZMO system variable. You can also suppress the display of gizmos when objects are selected.

After the gizmo is active, you can also switch to a different type of gizmo. The switching behavior differs, depending on when you select the objects:

- **Select objects first.** If a gizmo operation is in progress, you can press the Spacebar repeatedly to cycle through the other gizmo types. When you switch gizmos this way, the gizmo activity is constrained to the originally selected axis or plane.

During a gizmo operation, you can also select a different gizmo type on the shortcut menu.

- **Run the command first.** When you start the 3D Move, 3D Rotate, or 3D Scale operation before selecting objects, the gizmo is placed at the center of the selection set. Use the Relocate Gizmo option on the shortcut menu to relocate the gizmo anywhere in 3D space. You can also choose a different type of gizmo on the shortcut menu.

Change the Gizmo Settings

The following settings affect the display of gizmos:

- **Default gizmo.** The DEFAULTGIZMO system variable specifies which gizmo is displayed by default when an object is selected in a view with a 3D visual style. You can turn off display of the gizmo. This setting is also available on the ribbon.
- **Default location.** The GTLOCATION system variable sets the default location of the gizmo. The gizmo can be displayed at the center of the selection set (default), or it can be positioned at the 0,0,0 coordinates of the current UCS.
- **Automatic display.** The GTAUTO system variable sets whether gizmos are displayed automatically whenever you select objects in a 3D view that is set to a 3D visual style (default). If you turn off this system variable, the grips are not displayed until the gizmos are active.
- **Conversion of move, rotate, and scale operations from 2D to 3D.** Turn on the GTDEFAULT system variable to start the 3DMOVE, 3DROTATE, or 3DSCALE command automatically when the MOVE, ROTATE, or SCALE command is started in a 3D view. This system variable is turned off by default.
- **Active status of subobject grips.** If you press Ctrl+click a subobject, the GRIPSUBOBJMODE system variable sets whether the subobject grips are active immediately. Setting subobject grips to be active upon selection helps you modify groups of mesh subobjects without selecting them again.

To specify which 3D gizmo is displayed by default when an object is selected

- Click Home tab ➤ Subobject panel ➤ Move Gizmo, Rotate Gizmo, or Scale Gizmo.

Command entry: At the Command prompt, enter **DEFAULTGIZMO**. Then enter **0** (Move Gizmo), **1** (Rotate Gizmo), or **2** (Scale Gizmo).

To suppress the display of the 3D gizmo when an object is selected

- Click Home tab ➤ Subobject panel ➤ No Gizmo.

Command entry: At the Command prompt, enter **DEFAULTGIZMO,** then enter **3** (No Gizmo).

To set the default location for gizmos

1. At the Command prompt, enter **gtlocation**.
2. Do one of the following:
   - Enter **1** to set the location at the geometric center of the selection set.
   - Enter **0** to set the location to overlap the UCS icon.
3. Press Enter.

To relocate a gizmo

1. Right-click the center box (base grip) of the gizmo. Click Relocate Gizmo.
2. Click in the drawing area to specify a new location.

To change the type of gizmo while moving, rotating, or scaling objects

1. Select the 3D objects that you want to move, rotate, or scale.
   - To modify the entire object, select the object.
   - To modify a subobject (face, edge, or vertex), press Ctrl+click the subobject. (You can limit the selection set by specifying a subobject selection filter on the shortcut menu.)
2. Hover the cursor over an axis handle on the gizmo until it turns yellow and the axis vector displays. Then click the axis handle.
3. Press the Spacebar to cycle through the gizmo types until the correct gizmo is displayed.

   When you change gizmos this way, the movement remains constrained to the selected axis.

NOTE You cannot use the Spacebar to change the type of gizmo when the 3DMOVE, 3DROTATE, or 3DSCALE command is active.

Shortcut menu: Right-click the gizmo ➤ Move, Rotate, or Scale

Quick Reference

Commands

3DMOVE

In a 3D view, displays the 3D Move gizmo to aid in moving 3D objects a specified distance in a specified direction.

3DROTATE

In a 3D view, displays the 3D Rotate gizmo to aid in revolving 3D objects around a base point.

3DSCALE

In a 3D view, displays the 3D Scale gizmo to aid in resizing 3D objects.

System Variables

DEFAULTGIZMO

Sets the 3D Move, 3D Rotate, or 3D Scale gizmo as the default during subobject selection.

GRIPSUBOBJMODE

Sets whether edge, face, or vertex grips are active upon initial selection.

GTAUTO

Controls whether 3D gizmos are automatically displayed when you select objects before you start a command in a viewport with a 3D visual style.

GTDEFAULT

Controls whether the 3D Move, 3D Rotate, or 3D Scale operation starts automatically when you start the MOVE, ROTATE, or SCALE command in a viewport with a 3D visual style.

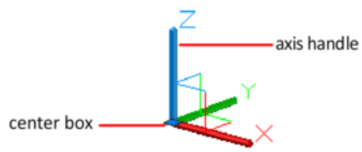
GTLOCATION

Controls the initial location of the 3D Move, 3D Rotate, or 3D Scale gizmo when you select objects before you start a command in a viewport with a 3D visual style.

Move 3D Objects

Move a selection set of objects and subobjects freely or constrain the movement to an axis or plane.

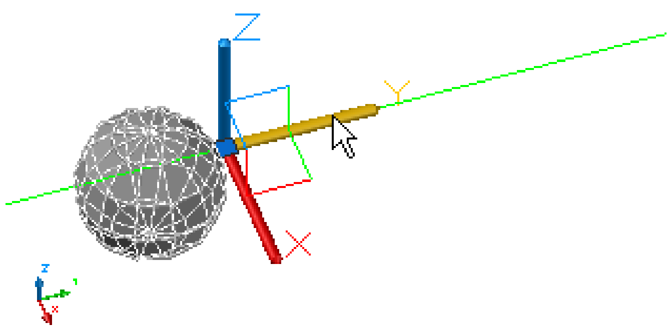
To move 3D objects and subobjects, click and drag the gizmo anywhere in 3D space. This location (indicated by the center box [or base grip] of the gizmo) sets the base point for the movement and temporarily changes the position of the UCS while you move the selected objects.



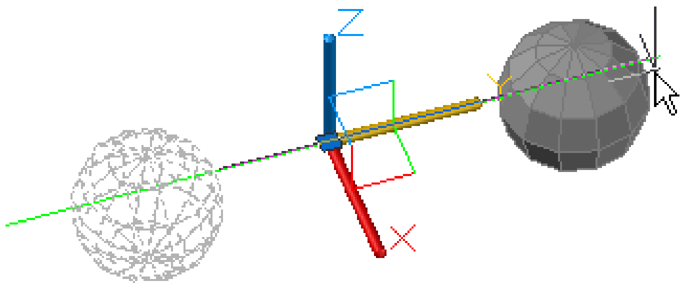
To move the objects freely, drag outside the gizmo or specify the axis or plane to which you will constrain the movement.

Constrain the Movement to an Axis

You can use the Move gizmo to constrain the movement to an axis. As the cursor hovers over an axis handle on the gizmo, a vector aligned with the axis is displayed, and the specified axis turns yellow. Click the axis handle.

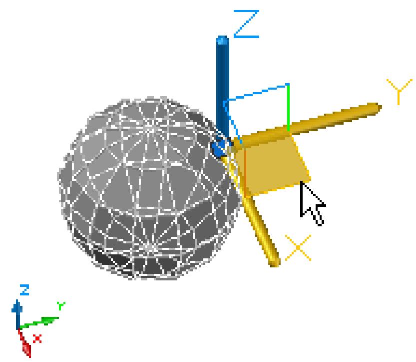


As you drag the cursor, movement of the selected objects and subobjects is constrained to the highlighted axis. You can click or enter a value to specify the distance of the move from the base point. If you enter a value, the movement direction of the object follows the initial direction of the cursor movement.

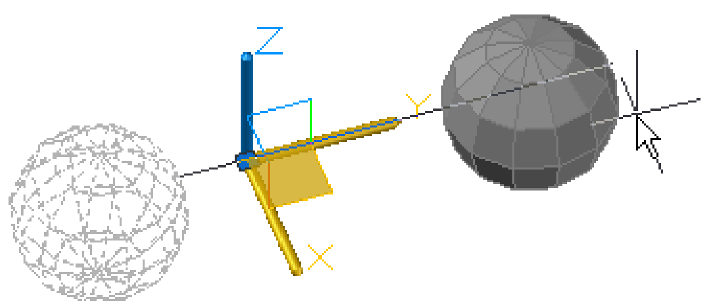


Constrain the Movement to a Plane

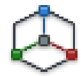
You can use the Move gizmo to constrain the movement to a plane. Each plane is identified by a rectangle that extends from the respective axis handles. You can specify the plane of movement by moving the cursor over the rectangle. When the rectangle turns yellow, click it.



As you drag the cursor, the selected objects and subobjects move only along the highlighted plane. Click or enter a value to specify the distance of the move from the base point.



To move objects in 3D space along a specified axis

1 Click Home tab ➤ Modify panel ➤ 3D Move. 

2 Select the objects and subobjects you want to move using the following methods:
   - Press and hold Ctrl to select subobjects (faces, edges, and vertices).
   - Release Ctrl to select entire objects.

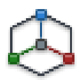
3 When you have selected all objects, press Enter.

   The Move gizmo is displayed attached to the cursor.

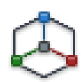
4 Click to place the Move gizmo, specifying the base point for the move.

5 Hover the cursor over an axis handle on the gizmo until it turns yellow and the vector is displayed. Then click the axis handle.

6 Click or enter a value to specify the distance of the move.

Toolbar: Modeling 
Command entry: 3DMOVE

To move objects in 3D space constrained to a specified plane

1 Click Home tab ➤ Modify panel ➤ 3D Move. 

2 Select the objects and subobjects you want to move using the following methods:
   - Press and hold Ctrl to select subobjects (faces, edges, and vertices).
   - Release Ctrl to select entire objects.

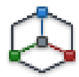
3 When you have selected all objects, press Enter.

   The Move gizmo is displayed attached to the cursor.

4 Click to place the Move gizmo, specifying the base point for the move.

5 Move the cursor over the planar rectangle that intersects the axis handles that define the plane of constraint. When the rectangle turns yellow, click it.

6 Click or enter a value to specify the distance of the move.

Toolbar: Modeling 
Command entry: 3DMOVE

Quick Reference

Commands

3DMOVE

In a 3D view, displays the 3D Move gizmo to aid in moving 3D objects a specified distance in a specified direction.

System Variables

DEFAULTGIZMO

Sets the 3D Move, 3D Rotate, or 3D Scale gizmo as the default during subobject selection.

Rotate 3D Objects

Constrain the rotation of 3D objects and subobjects to an axis.

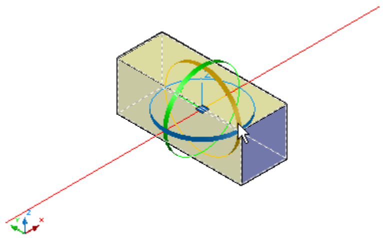
After you select the objects and subobjects that you want to rotate, the gizmo is located at the center of the selection set. This location is indicated by the center box (or base grip) of the gizmo. It sets the base point for the movement and temporarily changes the position of the UCS while you rotate the selected objects.

You then rotate the objects freely by dragging outside the gizmo. You can also specify an axis about which to constrain the rotation.

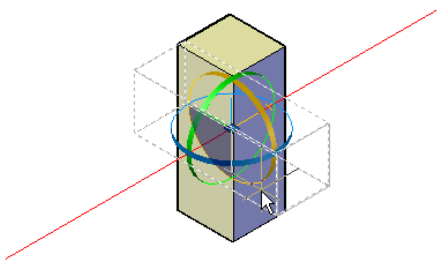
If you want to realign the center of rotation, you can relocate the gizmo by using the Relocate Gizmo option on the shortcut menu.

Constrain the Rotation to an Axis

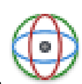
You can constrain the rotation to a specified axis. As you move the cursor over the rotation paths on the 3D rotate gizmo, a vector line representing the axis of rotation is displayed. Specify an axis of rotation by clicking the rotation path when it turns yellow.



When you drag the cursor, the selected objects and subobjects rotate about the base point along the specified axis. The gizmo displays the degree of rotation from the original position of the object as the object moves. You can click or enter a value to specify the angle of the rotation.



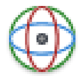
To rotate objects in 3D space along a specified axis

1. Click Home tab ➤ Modify panel ➤ 3D Rotate. 
2. Select the objects and subobjects you want to rotate using the following methods:
   - Press and hold Ctrl to select subobjects (faces, edges, and vertices).
   - Release Ctrl to select entire objects.
3. When you have selected all objects, press Enter.

   The rotate gizmo is displayed attached to the cursor.
4. Click to place the rotate gizmo, specifying the base point for the move.

5 Hover the cursor over an axis path on the gizmo until it turns yellow and the vector that represents the axis of rotation is displayed. Click the path.

6 Click or enter a value to specify the angle of the rotation.

Toolbar: Modeling 
Command entry: 3DROTATE

Quick Reference

Commands

3DROTATE

In a 3D view, displays the 3D Rotate gizmo to aid in revolving 3D objects around a base point.

System Variables

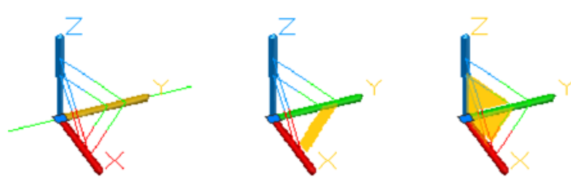
DEFAULTGIZMO

Sets the 3D Move, 3D Rotate, or 3D Scale gizmo as the default during subobject selection.

Scale 3D Objects

Change the size of 3D objects uniformly or along a specified axis or plane.

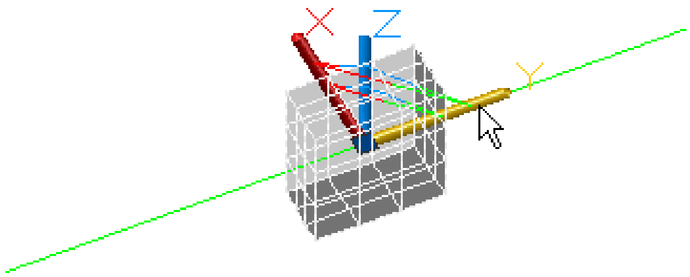
After you select the objects and subobjects to scale, you can constrain the object scaling by clicking the gizmo axis, plane, or the portion of the gizmo between all three axes.

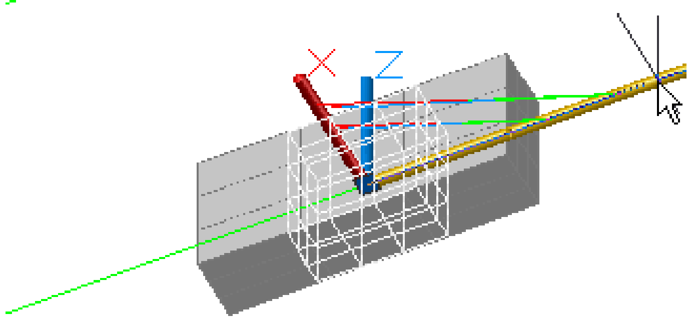


axis scale planar scale uniform scale

Scale a 3D Object Along an Axis

You can constrain object scaling to a specified axis. As you move the cursor over the axes on the 3D Scale gizmo, a vector line representing the axis of scale is displayed. Specify an axis of scale by clicking the axis when it turns yellow.

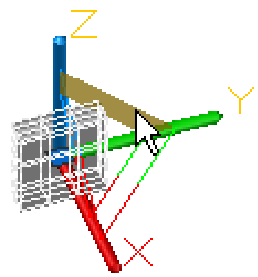


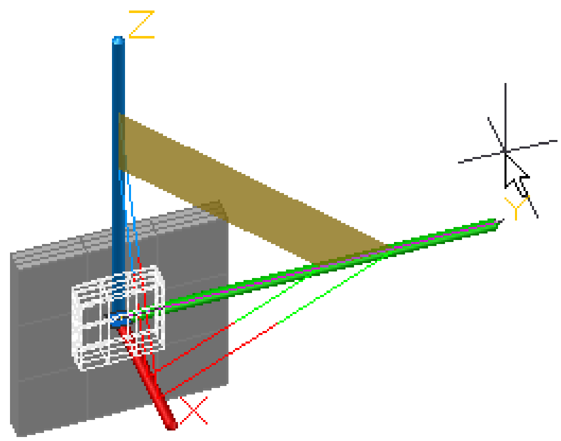


When you drag the cursor, the selected objects and subobjects are resized along the specified axis. You can click or enter a value to specify the scale from the selected base point.

Scale a 3D Object Along a Plane

You can constrain the object scaling to a specified plane. Each plane is identified by a bar that extends from the outer ends of the respective axis handles. You can specify the plane of scale by moving the cursor over one of the bars. When the bar turns yellow, click it.

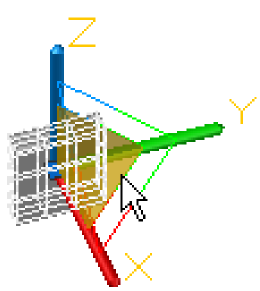


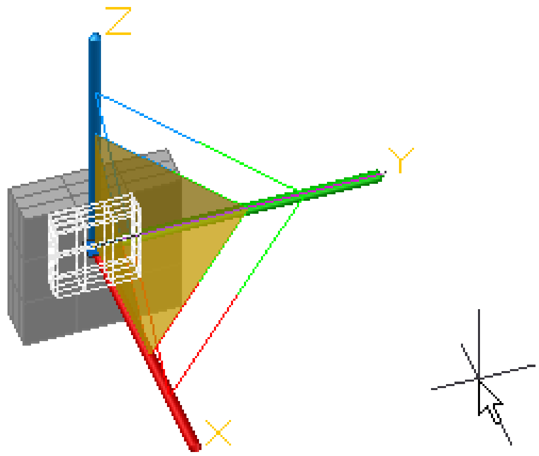


As you drag the cursor, the selected objects and subobjects are scaled only along the highlighted plane. Click or enter a value to specify the scale from the selected base point.

Scale a 3D Object Uniformly

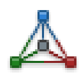
You can scale an object uniformly along all axes. As you move the cursor toward the center point of the gizmo, a highlighted triangular area indicates that you can click to scale the selected objects and subobjects along all three axes.





As you drag the cursor, the selected objects and subobjects are scaled uniformly. Click or enter a value to specify the scale from the selected base point.

To scale a 3D object along a specified axis

1 Click Home tab ➤ Modify ➤ 3D Scale. 

2 Select the objects and subobjects you want to scale using the following methods:

- Press and hold Ctrl to select subobjects (faces, edges, and vertices).
- Release Ctrl to select entire objects.

3 When you have selected all objects, press Enter.

The Scale gizmo is displayed at the center of the selected object or objects.

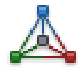
4 Specify a base point for the scale.

5 Hover the cursor over one of the axes of the gizmo until it turns yellow. Click the yellow axis.

6 Click or enter a value to specify the scale of the selected object.

Command entry: 3DSCALE

To scale a 3D object along a specified plane

1 Click Home tab ➤ Modify ➤ 3D Scale. 

2 Select the objects and subobjects you want to scale using the following methods:

   - Press and hold Ctrl to select subobjects (faces, edges, and vertices).
   - Release Ctrl to select entire objects.

3 When you have selected all objects, press Enter.

   The Scale gizmo is displayed at the center of the selected object or objects.

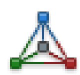
4 Specify a base point for the scale.

5 Hover the cursor over one of the bars found between each of the axes of the gizmo until it turns yellow. Click the yellow bar.

6 Click or enter a value to specify the scale of the selected object.

Command entry: 3DSCALE

To scale a 3D object uniformly

1 Click Home tab ➤ Modify ➤ 3D Scale. 

2 Select the objects and subobjects you want to scale using the following methods:

   - Press and hold Ctrl to select subobjects (faces, edges, and vertices).
   - Release Ctrl to select entire objects.

3 When you have selected all objects, press Enter.

   The Scale gizmo is displayed at the center of the selected object or objects.

4 Specify a base point for the scale.

5 Hover the cursor over the triangular area closest to the center point of the gizmo until it turns yellow. Click the yellow area.

6 Click or enter a value to specify the scale of the selected object.

Command entry: 3DSCALE

Quick Reference

Commands

3DSCALE

In a 3D view, displays the 3D Scale gizmo to aid in resizing 3D objects.

System Variables

DEFAULTGIZMO

Sets the 3D Move, 3D Rotate, or 3D Scale gizmo as the default during subobject selection.

Modify 3D Solid Subobjects

Modify the shape of a 3D solid by editing its subobjects (faces, edges, and vertices).

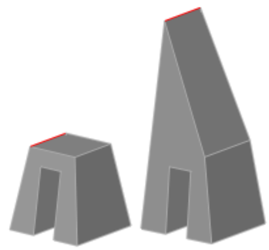
Move, Rotate, and Scale 3D Subobjects

Move, rotate, and scale individual subobjects on 3D solids.

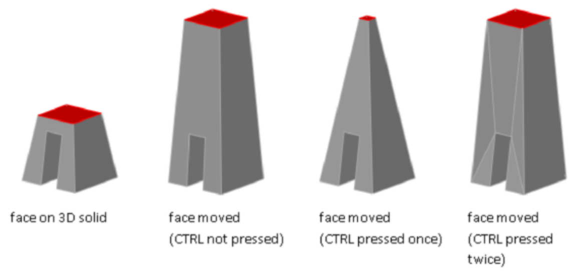
Use the same methods to modify a face, edge, or vertex that you use to modify the entire object:

- Drag grips
- Use gizmos (3DMOVE, 3DROTATE, and 3DSCALE)
- Enter object editing commands (MOVE, ROTATE, and SCALE)

When you move, rotate, or scale a subobject, the subobject is modified in a way that maintains the integrity of the 3D solid. For example, when you drag an edge to move it, the adjacent faces are adjusted so that they remain adjacent to the edge.



Several results are possible when you modify a solid. When you move, rotate, or scale subobjects, you can press Ctrl one or more times as you drag to cycle through modification options. The following illustration shows the modification options for moving a face.



Move, Rotate, and Scale Subobjects on Composite Solids

When you modify composite solids, the effect of the edits depends on the current setting of the History property.

- To modify subobjects of each history component separately, the History property must be set to Record.
- To modify subobjects of the combined composite solid as a whole, the History property must be set to None.

Rules and Limitations When Moving, Rotating, and Scaling Subobjects

You can only move, rotate, and scale subobjects on 3D solids if the operation maintains the integrity of the solid. The following rules and limitations apply to moving, rotating, and scaling subobjects:

- When you use grips to modify subobjects, grips are not displayed on the subobjects that cannot be moved, rotated, or scaled.
- In most cases, you can move, rotate, and scale both planar and non-planar faces.

- You can only modify an edge that is a straight line and that has at least one planar adjacent face. The planes of the adjacent planar faces are adjusted to contain the modified edge.
- You cannot move, rotate, or scale edges (or their vertices) that are imprinted inside faces.
- You can only modify a vertex if it has at least one planar adjacent face. The planes of the adjacent planar faces are adjusted to contain the modified vertex.
- When you drag a subobject, the final result might be different than the preview displayed during the modification. This result occurs when the solid geometry is adjusted in order to maintain its topology. In some cases, the modification is not possible because it changes the topology of the solid too severely.
- If the modification causes spline surfaces to be extended, the operation is often unsuccessful.
- You cannot move, rotate, or scale non-manifold edges (edges that are shared by more than two faces) or non-manifold vertices. Also, if some non-manifold edges or vertices are present near faces, edges, and vertices that you modify, the operation might not be possible.

See also:

- Modify Mesh Objects on page 1337

Quick Reference

Commands

3DMOVE

In a 3D view, displays the 3D Move gizmo to aid in moving 3D objects a specified distance in a specified direction.

3DROTATE

In a 3D view, displays the 3D Rotate gizmo to aid in revolving 3D objects around a base point.

3DSCALE

In a 3D view, displays the 3D Scale gizmo to aid in resizing 3D objects.

MOVE

Moves objects a specified distance in a specified direction.

ROTATE

Rotates objects around a base point.

SCALE

Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling.

System Variables

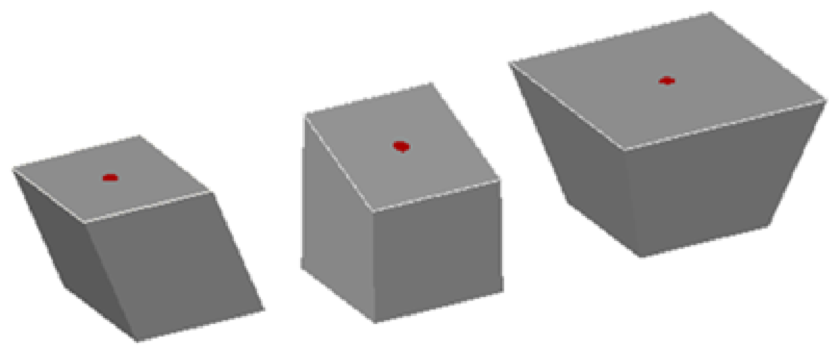
No entries

Modify Faces on 3D Objects

Modify individual faces on 3D objects using a variety of methods.

Move, Rotate, and Scale Faces on 3D Solids

Modify the location, rotation, and size of faces on a 3D solids.



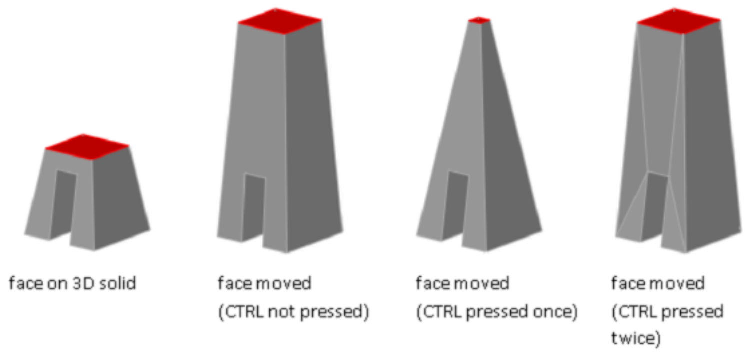
cube with top face moved, rotated, and scaled

Use the MOVE, ROTATE, and SCALE commands to modify faces just as you would with any other object. Press Ctrl+click to select a face on a solid.

If you move, rotate, or scale a face on a 3D solid primitive, the solid primitive's history is removed. The solid is no longer a true primitive and cannot be manipulated using grips or the Properties palette.

Face Modification Options

As you drag a face, press Ctrl to cycle through modification options.



- **Maintain shape of face, modify adjacent faces.** When you move or rotate a face without pressing Ctrl, the shape and size of the face is maintained. However, the planes of adjacent faces might change.
- **Modify shape of face, retain edges.** When you move or rotate a face and press and release Ctrl once while dragging, the size of the face is modified within the boundary, or footprint, of the adjacent faces.
- **Modify face, triangulate adjacent faces.** When you move or rotate a face and press and release Ctrl twice while dragging, the size and shape of the face is maintained. (This behavior is the same as if you had not pressed Ctrl). However, the adjacent planar faces are triangulated (divided into two or more planar triangular faces), if necessary.

If you press and release Ctrl a third time, the modification returns to the first option, as if you had not pressed Ctrl.

Quick Reference

Commands

MOVE

Moves objects a specified distance in a specified direction.

ROTATE

Rotates objects around a base point.

SCALE

Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling.

System Variables

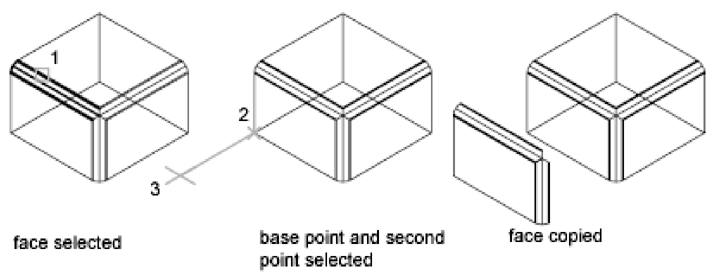
No entries

Copy, Delete, and Color Faces on 3D Solids

Copy, remove, or change the color of faces on 3D solid objects.

Copy a Face

You can duplicate the face of a 3D solid object using the copy option of the SOLIDEDIT command. The selected faces are copied as regions or bodies.

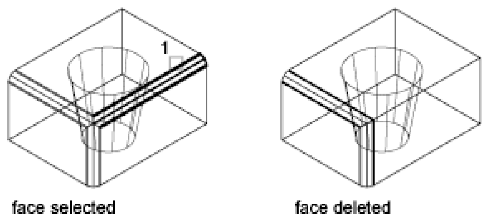


Delete a Face

If you specify two points, the first point is used as a base point and a single copy is placed relative to the base point. If you specify a single point and press Enter, the original selection point is used as a base point. The next point is the point of displacement.

If a face is surrounded by coplanar faces, you can delete it using the following methods:

- Select the face and press Delete.
- Select the face and enter **erase**.
- Use the Delete option of the SOLIDEDIT command.



Color a Faces

You can modify the color of a face on a 3D solid by selecting the face and then changing the Color property in the Properties palette.

See also:

- Split or Extrude Mesh Faces on page 1352

To copy a face on a solid object

1. Click Home tab ➤ Solid Editing panel ➤ Face Editing drop-down ➤ Copy Faces. 
2. Select the face to copy.
3. Select additional faces or press Enter to copy.
4. Specify the base point for the copy.
5. Specify the second point of displacement and press Enter.

NOTE Use EXTRUDE to extrude a copied face.

Command entry: SOLIDEDIT

To delete a face on a solid object

1. Click Home tab ➤ Solid Editing panel ➤ Face Editing drop-down ➤ Delete Faces.
2. Select the face to delete. (It must be surrounded by faces that share the same plane.)
3. Select additional faces or press Enter to delete.
4. Press Enter to complete the command.

Toolbar: Solid Editing
Command entry: SOLIDEDIT

To change the color of a face on a solid object

1. Press and hold Ctrl as you click a face on a 3D solid.
2. If the Properties palette is not displayed, select any object. Right-click the object to display the shortcut menu. Click Properties.
3. In the Properties palette, under General, click the Color arrow and select a color from the list

   For additional color options, click Select Color to display the Select Color dialog box. Specify a color and click OK.

   PROPERTIES

Quick Reference

Commands

PROPERTIES

Controls properties of existing objects.

SOLIDEDIT

Edits faces and edges of 3D solid objects.

System Variables

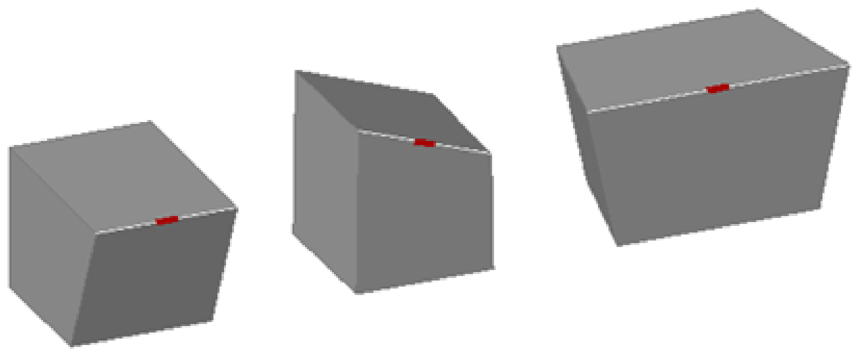
No entries

Modify Edges on 3D Objects

You can select and modify edges on a 3D solid.

Move, Rotate, and Scale Edges

Move, rotate, and scale the edges on 3D solids using grips, gizmos, and commands.



cubes with edges moved, rotated, and scaled

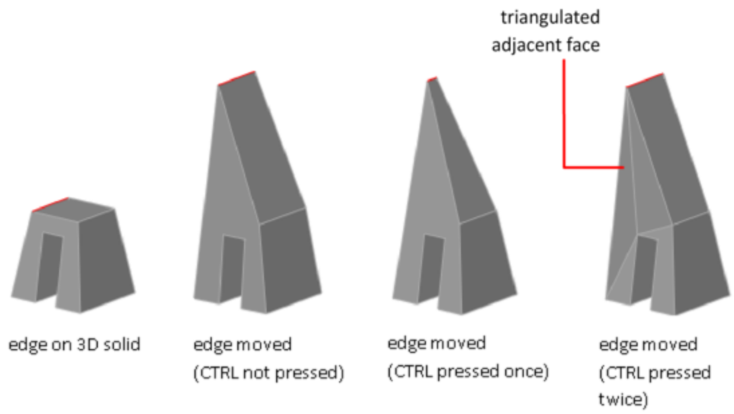
You can use MOVE, ROTATE, and SCALE to modify edges on 3D solids just as you can for any other object. Press and hold Ctrl to select the edge.

If you move, rotate, or scale an edge on a 3D solid primitive, the history of the solid primitive is removed. The solid is no longer a true primitive and cannot be manipulated using grips and the Properties palette.

Edges on regions can be selected, but do not display grips. These edges can also be moved, rotated, and scaled.

Edge Modification Options

As you drag an edge, press Ctrl to cycle through modification options.



- **Maintain length of edge.** When you move, rotate, or scale an edge without pressing Ctrl, the shared length of the edge and its vertices is maintained. However, the planes of the adjacent faces adjacent might be changed.
- **Change the length of the edge.** When you move, rotate, or scale an edge and press and release Ctrl once while dragging, the edge is modified without its vertices. The surfaces of the adjacent faces are maintained, but the length of the modified edge might change.
- **Triangulate adjacent faces.** When you move, rotate, or scale an edge and press and release Ctrl twice while dragging, the edge and its vertices are modified. (This behavior is the same as if you had not pressed Ctrl). However, if the adjacent faces are no longer planar, they are triangulated (divided into two or more planar triangular faces).

If you press and release Ctrl a third time, the modification returns to the first option, as if you had not pressed Ctrl.

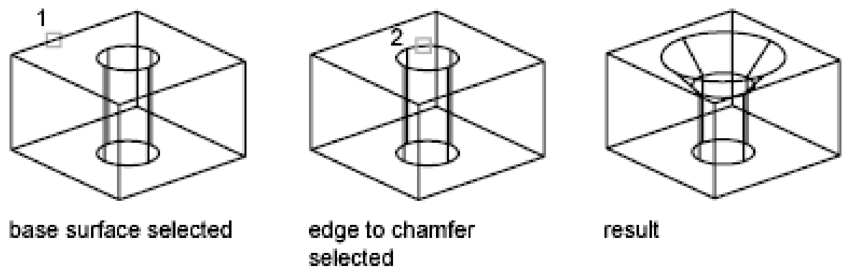
Delete Edges

You can also delete edges that completely divide two coplanar faces using one of the following methods:

- Select the edge and press Delete.
- Select the edge and enter the ERASE command.

Fillet and Chamfer 3D Solids and Surfaces

Add rounds and bevels to selected 3D solids using FILLET and CHAMFER. You can modify the properties of the resulting 3D subobjects in the Properties palette.



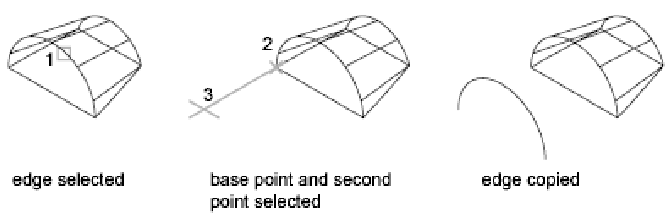
With the default method, you can specify the fillet radius and then select the edges to fillet. Alternatively, you can specify individual measurements for each filleted edge and fillet a tangential series of edges.

Color Edges

You can modify the color of an edge on a 3D object by selecting the edge and changing the Color property in the Properties palette.

Copy Edges

You can copy individual edges on a 3D solid object. Edges are copied as lines, arcs, circles, ellipses, or splines.



If you specify two points, the first point is used as a base point and a single copy is placed relative to the base point. If you specify a single point, and then press Enter, the original selection point is used as a base point. The next point is used as a point of displacement.

See also:

- Move, Rotate, and Scale 3D Subobjects on page 1310

- Use Gizmos to Modify Objects on page 1294
- Modify Objects on page 1091
- Move or Rotate Objects on page 1093
- Resize or Reshape Objects on page 1122
- Overview of Modifying Meshes on page 1337

To fillet a solid object

1. Click Home tab ➤ Modify panel ➤ Fillet.
2. Select the edge of the solid to fillet.
3. Specify the fillet radius.
4. Select additional edges or press Enter to fillet.

Command entry: FILLET

To chamfer a 3D solid object

1. Click Home tab ➤ Modify panel ➤ Chamfer.
2. Select the edge of the base surface to chamfer.

   One of two surfaces adjacent to the selected edge is highlighted.
3. Do one of the following:
   - To select a different surface, enter **n** (Next).
   - To use the current surface, press Enter.
4. Specify the base surface distance.

   The base surface distance is measured from the selected edge to a point on the base surface. The other surface distance is measured from the selected edge to a point on the adjacent surface.
5. Specify the location of the chamfer using one of the following options.
   - To specify an individual edge, select the edge.

- To select all edges around the base surface, enter **l** (Loop). Specify an edge.

6 To complete the chamfer, press Enter.

Command entry: CHAMFER

To modify a fillet or chamfer on a 3D solid

1 Press and hold Ctrl as you select a fillet or chamfer on a 3D solid.

2 If the Properties palette is not displayed, select any object. Right-click the object to display the shortcut menu. Click Properties.

3 In the Properties palette, modify the properties of the fillet or chamfer.

Command entry: PROPERTIES

To change the color of an edge on a solid object

1 Press and hold Ctrl as you click an edge on a 3D solid.

2 If the Properties palette is not displayed, select any object. Right-click the object to display the shortcut menu. Click Properties.

3 In the Properties palette, under General, click the Color arrow and select a color from the list

For additional color options, click Select Color to display the Select Color dialog box. Specify a color and click OK.

PROPERTIES

To copy an edge on a solid object

1 Click Home tab ➤ Solid Editing panel ➤ Edge Editing drop-down ➤ Copy Edges.

2 Press Ctrl+click the edge of the face to copy.

3 Select additional edges, if needed, and press Enter.

4 Specify the base point of the copied edges.

5 Specify the second point of displacement to indicate the location of the copied edges.

6 Press Enter to complete the command.

Toolbar: Solid Editing
Command entry: SOLIDEDIT

Quick Reference

Commands

EDGE

Changes the visibility of 3D face edges.

CHAMFER

Bevels the edges of objects.

FILLET

Rounds and fillets the edges of objects.

MOVE

Moves objects a specified distance in a specified direction.

PROPERTIES

Controls properties of existing objects.

ROTATE

Rotates objects around a base point.

SCALE

Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling.

SOLIDEDIT

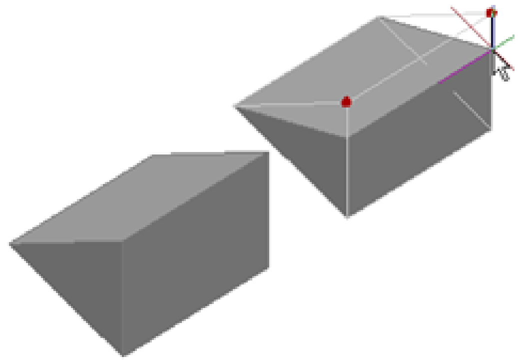
Edits faces and edges of 3D solid objects.

System Variables

No entries

Modify Vertices on 3D Objects

Move, rotate, scale, or drag the vertices of 3D solids.



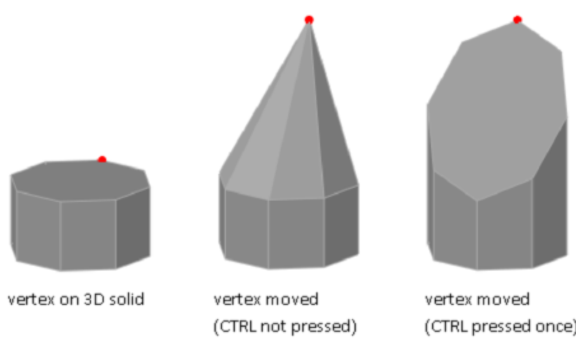
wedge with two vertices moved

You can modify the form of a 3D solid by modifying one or more vertices. Use grips and gizmos, or run the MOVE, ROTATE, or SCALE command. When you scale or rotate vertices, you must select two or more vertices to see a change in the solid object. Clicking and dragging a vertex “stretches” the 3D object.

If you move, rotate, or scale one or more vertices on a 3D solid primitive, the solid primitive history is removed. The solid is no longer a true primitive and cannot be modified using grips and the Properties palette.

Vertex Modification Options

As you drag a vertex, press Ctrl to cycle through modification options.



- **Triangulate adjacent faces.** When you move, rotate, or scale a vertex without pressing Ctrl, some adjacent planar faces may be triangulated (divided into two or more planar triangular faces).

- **Modify some adjacent faces without triangulation.** When you move, rotate, or scale a vertex and press and release Ctrl once, some adjacent planar faces might be adjusted.

If you press and release Ctrl a second time, the modification returns to the first option, as if you had not pressed Ctrl.

Delete a Vertex

You can delete a vertex that connects two parallel edges that are collinear and do not intersect on any other edges.

See also:

- Move, Rotate, and Scale 3D Subobjects on page 1310
- Use Gizmos to Modify Objects on page 1294
- Modify Objects on page 1091
- Overview of Modifying Meshes on page 1337

To move a vertex on a 3D object

1. Click Home tab ➤ Subobject panel ➤ Vertex.
2. Press Ctrl+click a vertex on a 3D object.
3. Drag the vertex to the required location.

To delete a vertex on a 3D object

1. Click Home tab ➤ Subobject panel ➤ Vertex.
2. Press Ctrl+click a vertex that connects collinear edges are parallel and do not lie on any other edges.
3. Press Delete.

Quick Reference

Commands

MOVE

Moves objects a specified distance in a specified direction.

ROTATE

Rotates objects around a base point.

SCALE

Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling.

SOLIDEDIT

Edits faces and edges of 3D solid objects.

System Variables

No entries

Work with Complex 3D Solids and Surfaces

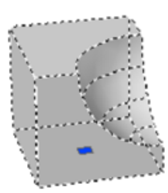
Modify composite solids that are created by a union, subtract, intersect, fillet, or chamfer process.

Display Original Forms of Composite Solids

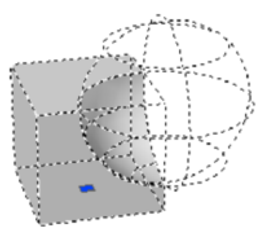
By default, 3D composite objects retain a history that displays an editable image of their original component forms.

Retain the History of the Composite Components

After you create a composite object, you can modify the shape of the new object by modifying a highlighted wireframe image of its original components. If the Show History property is Yes, wireframes of the original forms—including forms that have been removed—are displayed in a dimmed state. (The SHOWHIST system variable also controls this setting.)



composite solid with Show History set to no



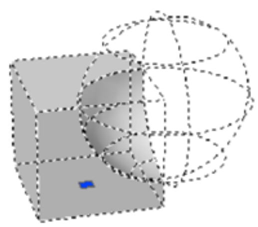
composite solid with Show History set to yes

To retain a history of the original parts of composite solids, the History property must be set to Record (Properties Palette) when the composite operation occurs. (You can also use the SOLIDHIST system variable to set this property.)

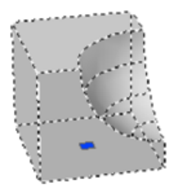
Display and Remove the History to Modify the Composite

When you modify the composite object, you can display the history. Then use the grips on the history subobject to modify the object. For more information about using grips with composite solids, see Modify Composite Solids and Surfaces on page 1328.

You can remove the history of a selected composite object by changing its History setting to None, or by entering the BREP command. After a history has been removed, you can no longer select and modify the original, removed, components of the solid. You can restart history retention for the solid by changing its History setting back to Record.



composite solid with Show History set to yes



composite solid with history removed and Show History set to yes

Removing a composite history is useful when you work with complex composite solids. After you create the initial complex form, set History to None to remove the history. Then reset the value to Record. With this process, you can create a complex composite object, and then reset it to serve as a base form for additional composite operations.

To display the original components of a composite solid

1 If the Properties palette is not displayed, select any object. Right-click the object to display the shortcut menu. Click Properties.

2 In a drawing, select a 3D composite solid.

3 On the Properties palette, Solid History area, under Show History, select Yes.

To remove the history of a solid object

1 If the Properties palette is not displayed, select any object. Right-click the object to display the shortcut menu. Click Properties.

2 In a drawing, select a 3D solid.

3 On the Properties palette, Solid History area, under History, select None.

Command entry: BREP

To set a 3D solid to record a history of its original forms

1 If the Properties palette is not displayed, select any object. Right-click the object to display the shortcut menu. Click Properties.

2 In a drawing, select a solid.

3 On the Properties palette, Solid History area, under History, select Record.

Quick Reference

Commands

BREP

Removes the history from 3D solid primitives and composite solids.

System Variables

SHOWHIST

Controls the Show History property for solids in a drawing.

SOLIDHIST

Controls the default History property setting for new and existing objects.

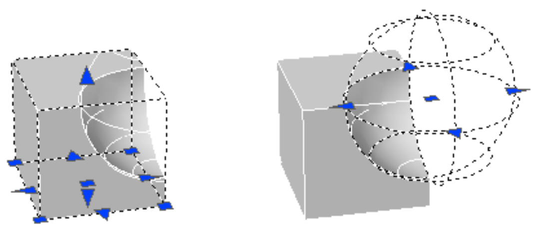
Modify Composite Solids and Surfaces

Modify the entire form of a composite 3D object or the original forms that make up the composite.

You can move, scale, or rotate a selected composite object using grips or gizmos.

Modify Original Components of Composites

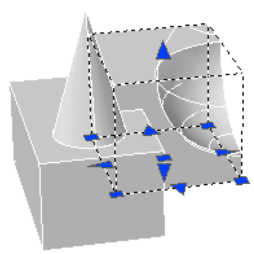
When the History property is set to Record, press the Ctrl key to display any original forms that were removed during a union, subtract, or intersect operation. If the original, removed form was a solid primitive, you can drag the displayed grips to change its shape and size. As a result, the composite object is modified.



If the selected individual form does not contain its history, you can move, rotate, scale, or delete the form.

Modify Complex Composites

A composite object might be made up of other composite objects. You can select the history images of composite objects by holding down the Ctrl key as you click the forms. (For best results, set the subobject selection filter to Solid History.)



You can also change the size and shape of composite objects by clicking and dragging grips on individual faces, edges, and vertices. For more information, see Modify 3D Solid Subobjects on page 1310.

Separate Discrete Objects Combined with a Union

If you have combined discrete 3D solids or surfaces using a union operation, you can separate them into their original components. (Use the Separate option of the SOLIDEDIT command.) To be separated, the composite objects cannot overlap or share a common area or volume.

After separation, the individual solids retain their original layers and colors. All nested 3D solid objects are restored to their simplest forms.

See also:

- Use Grips to Edit Objects on page 1141
- Overview of Modifying Meshes on page 1337

To select an individual solid that is part of a composite solid

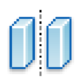
- Press Ctrl + click an individual solid that is part of a composite solid.

To separate a 3D solid composite into individual solids

1. Click Home tab ➤ Solid Editing panel ➤ Solid Editing drop-down ➤ Separate.
2. Select the 3D solid object.

   NOTE This operation only applies to non-intersecting objects that have been combined by union.

3. Press Enter to complete the command.

Toolbar: Solid Editing 
Command entry: SOLIDEDIT

Quick Reference

Commands

SOLIDEDIT

Edits faces and edges of 3D solid objects.

System Variables

LEGACYCTRLPICK

Specifies the keys for selection cycling and the behavior for Ctrl+click.

Shell and Remove Redundancies in 3D Objects

Convert 3D solids to shells and remove redundant lines and edges.

Shell 3D Solids

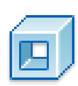
Convert a 3D solid to a hollow wall, or shell.

When you can convert a solid object to a shell, new faces are created by offsetting existing faces inside or outside their original positions.

Continuously tangent faces are treated as a single face when they are offset.

To create a 3D solid shell

1. Click Home tab ➤ Solid Editing panel ➤ Solid Editing drop-down ➤ Shell.
2. Select the 3D solid object.
3. Select one or more faces to be excluded from shelling.
4. Press Enter.
5. Specify the shell offset value.
   A positive offset value creates a shell wall in a positive direction from the face. A negative value creates a shell wall in a negative direction from the face.
6. Press Enter to complete the command.

Toolbar: Solid Editing 
Command entry: SOLIDEDIT

Quick Reference

Commands

SOLIDEDIT

Edits faces and edges of 3D solid objects.

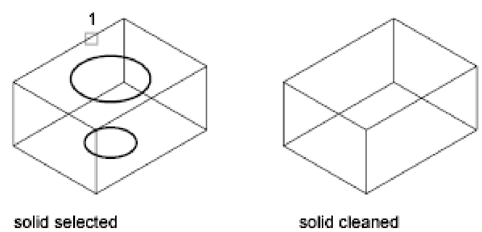
System Variables

No entries

Clean and Check 3D Solids

Remove redundant faces, edges, and vertices from a 3D solid, and verify whether the 3D solid is valid.

You can remove redundant edges or vertices that share the same surface or vertex definition. This operation merges adjacent faces and deletes all redundant edges, including imprinted and unused edges.



True 3D solid objects have editable properties, volume, and mass that are not shared by objects created with thickness or closed surfaces. You can check whether an object is a valid 3D solid by verifying whether it is listed as “3D Solid” on the Properties palette. You can also use SOLIDEDIT to verify whether a solid object is a valid 3D solid object.

To remove redundant lines from a 3D solid object

1. Click Home tab ➤ Solid Editing panel ➤ Solid Editing flyout ➤ Clean.
2. Select the 3D solid object.
3. Press Enter to complete the command.

Toolbar: Solid Editing 
Command entry: SOLIDEDIT

To validate a 3D solid object

1. Click Home tab ➤ Solid Editing panel ➤ Solid Editing flyout ➤ Check.
2. Select the 3D solid object.
3. Press Enter to complete the command.
   If the object is a valid 3D object, a message is displayed at the Command prompt. If it is not valid, you continue to be prompted to select a 3D solid.

Toolbar: Solid Editing 
Command entry: SOLIDEDIT

Quick Reference

Commands

SOLIDEDIT

Edits faces and edges of 3D solid objects.

System Variables

SOLIDCHECK

Turns 3D solid validation on and off for the current session.

Press or Pull Bounded Areas

Create a positive or negative extrusion in the shape of a bounded area.

bounded area (circle) on solid | bounded area pressed | bounded area pulled

Press in or pull out bounded, or closed, areas to create 3D holes and positive extrusions.

object formed using press and pull operations on an imprinted pyramid

In combination with imprinted faces, you can form complex shapes using press or pull operations to create extrusions and notches.

Methods for Press and Pull Modifications

With the PRESSPULL command, you specify the area to be extruded, and then move the cursor or enter a value to specify the length of the extrusion. The result is a single 3D solid object, often with a composite shape.

You can also press Ctrl+Shift+E to initiate a press or pull operation. To limit the type of objects that can act as boundaries, turn off the IMPLIEDFACE system variable. When the variable is off, only 3D faces and 3D solid faces can be extruded using Ctrl+Shift+E. (This variable does not affect the PRESSPULL command.)

NOTE If you alternatively use EXTRUDE to extend an existing face on a 3D solid, a separate extruded object is created.

Types of Objects That Can Be Pressed or Pulled

You can press or pull several types of bounded areas, including closed objects, areas enclosed by coplanar geometry, the faces of 3D solids, and an imprinted area on the face of a 3D solid. For a complete list of objects that are eligible for a press or pull extrusion, see PRESSPULL.

You cannot taper the pressed or pulled shape as you create it. However, you can achieve the same effect later by modifying the edges of the bounded area.

To press or pull a bounded area

1 Press and hold Ctrl+Shift+E.

2 Click any area bounded by coplanar lines or edges.

3 Drag the mouse to press or pull the bounded area.

4 Click two points or enter a value to specify the height or depth of the extrusion.

Toolbar: Modeling 
Command entry: PRESSPULL

Quick Reference

Commands

EXTRUDE

Extends the dimensions of a 2D object or 3D face into 3D space.

PRESSPULL

Presses or pulls bounded areas.

System Variables

IMPLIEDFACE

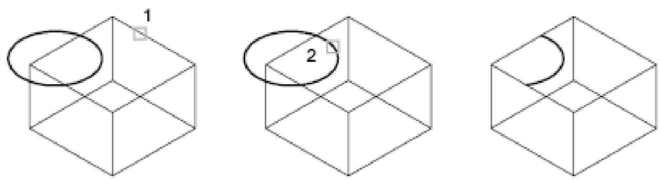
Controls the detection of implied faces.

Add Edges and Faces to Solids and Surfaces

Add editable faces to 3D solids and surfaces by imprinting other objects, such as arcs and circles.

With the IMPRINT command, you can add a new face to a 3D solid or surface by imprinting a coplanar object that overlaps the selected face. Imprinting provides additional edges that you can modify to reshape the solid object.

For example, if a circle overlaps the face of a box, you can imprint the intersecting curves on the solid.



You can delete or retain the original object as you imprint it.

Objects that can be imprinted on 3D solids and surfaces include arcs, circles, lines, 2D and 3D polylines, ellipses, splines, regions, bodies, and other 3D solids.

Edit Imprinted Objects

You can edit imprinted objects and subobjects in many of the same ways that you can edit other faces. For example, you can Ctrl+click to select a new edge and then drag to change its location.

The following limitations exist for imprinted objects:

- You can move the edges of the imprinted face only within the plane of a face.
- You might be unable to move, rotate, or scale some subobjects.
- Imprinted edges and faces might be lost when some subobjects are moved, rotated, or scaled.

Subobjects with editing limitations include

- Faces with imprinted edges or faces
- Edges or vertices with adjacent faces that contain imprinted edges or faces

To imprint a 3D solid object

1 Click Home tab ➤ Solid Editing panel ➤ Edge Editing drop-down ➤ Imprint.

2 Select the 3D solid object.

3 Select a co-planar object that you want to imprint.

4 Press Enter to retain the original objects, or enter **y** to delete them.

5 Select additional objects to imprint or press Enter.

6 Press Enter to complete the command.

Toolbar: Solid Editing 
Command entry: IMPRINT

Quick Reference

Commands

IMPRINT

Imprints 2D geometry on a 3D solid or surface, creating additional edges on planar faces.

System Variables

No entries

Modify Mesh Objects

Model mesh objects by changing smoothing levels, refining specific areas, or adding creases.

Overview of Modifying Meshes

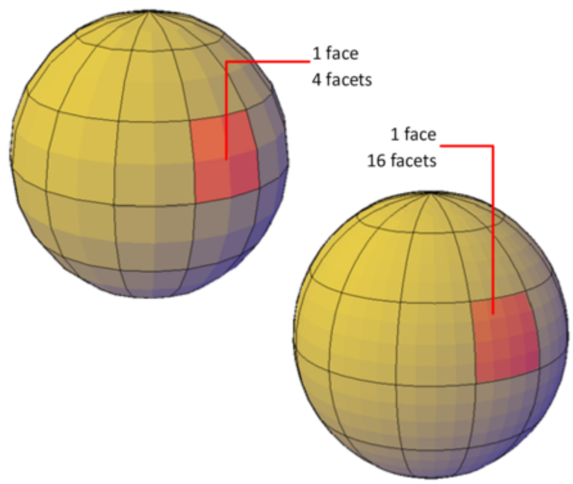
Modeling mesh objects differs from modeling 3D solids and surfaces in some important ways.

Mesh objects do not have the mass and volume properties of 3D solids. However, they do offer unique capabilities that enable you to design less angular, more rounded models.

NOTE The capabilities described in this section apply only to mesh objects created in AutoCAD 2010 and later. They cannot be used with legacy polyface or polygon mesh.

About Mesh Faces

Mesh objects are composed of faces and facets.



Faces are non-overlapping units that—along with their edges and vertices—form the basic editable units of a mesh object. When you move, rotate, and scale individual mesh faces, surrounding faces are stretched and deformed in order to avoid introducing gaps. When gaps occur, you can often close them by smoothing the object or refining individual faces.

About Mesh Facets

Mesh faces have underlying structures, known as facets. The density of the facet grid corresponds to the smoothness of the mesh. As the smoothness level is increased, the density of the underlying facet grid also increases. When you want to confine detailed mesh editing to a smaller area, you can convert facets to editable faces by using refinement.

Unlike faces, facets cannot be individually modified. However, you can make them more visible by modifying the VSLIGHTINGQUALITY system variable.

About Mesh Modeling

You can work with mesh objects in the following ways:

- **Add smoothness.** Increase or decrease smoothness levels to round the overall shape of the model. The underlying density of the mesh facet grid increases as the mesh object smoothness level increases.
- **Refine the object to reset the baseline smoothness level.** Refine a mesh object to convert the underlying facet grid to editable faces. Refinement also resets the lowest level of smoothness that can be applied to the object.

- **Refine a face.** Restrict the refinement to a specific mesh face. This method avoids resetting the smoothness baseline.
- **Crease an edge.** Remove the smoothness from specified edges. You can also remove an existing crease.
- **Split a face.** Divide an existing face into separate components along a path you specify.
- **Extrude a face.** Deform a specified face through extrusion. Unlike 3D solid extrusion, a mesh extrusion does not create a separate object.

Use Grip Editing with Mesh

Grips, as described in Use Grips to Edit 3D Solids and Surfaces on page 1291, are not available with meshes. However, you can manipulate the entire mesh model or individual subobjects using the following methods:

- **Subobject selection and editing.** Select faces, edges, and vertices the same way you select 3D solid subobjects. Press Ctrl and click the component. The subobject highlighting indicates what is selected. Press Shift and click again to remove the selection from a subobject. By turning on the Subobject Selection Filter, you can restrict selection to a specific subobject. See Select 3D Subobjects on page 1285.
- **Gizmo editing**. When you select a mesh object or subobject, the 3D Move, Rotate, or Scale gizmo is displayed automatically. (You can set which gizmo is displayed by default.) Use these gizmos to modify the selection uniformly, or along a specified plane or axis. See Use Gizmos to Modify Objects on page 1294.

Because dense meshes can be difficult to work with, you can change settings to improve the display and behavior of grips.

- **Set the subobject selection filter to select only faces, edges, or vertices:** Set the DEFAULTGIZMO system variable or use the shortcut menu.
- **Set whether a grip on a face, edge, or vertex is active immediately when you select the subobject:** Set the GRIPSUBOBJMODE system variable.

Quick Reference

Commands

EXTRUDE

Extends the dimensions of a 2D object or 3D face into 3D space.

MESHCREASE

Sharpens the edges of selected mesh subobjects.

MESHREFINE

Multiplies the number of faces in selected mesh objects or faces.

MESHSMOOTHLESS

Decreases the level of smoothness for mesh objects by one level.

MESHSMOOTHMORE

Increases the level of smoothness for mesh objects by one level.

MESHSPLIT

Splits a mesh face into two faces.

MESHUNCREASE

Removes the crease from selected mesh faces, edges, or vertices.

System Variables

DEFAULTGIZMO

Sets the 3D Move, 3D Rotate, or 3D Scale gizmo as the default during subobject selection.

GRIPSUBOBJMODE

Sets whether edge, face, or vertex grips are active upon initial selection.

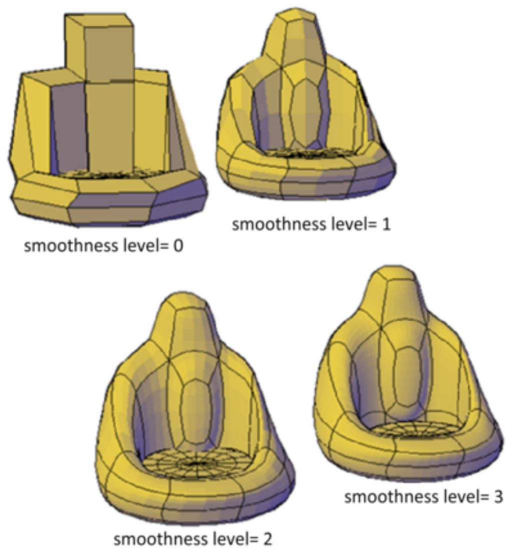
VSLIGHTINGQUALITY

Sets the lighting quality in the current viewport.

Change Mesh Smoothness Levels

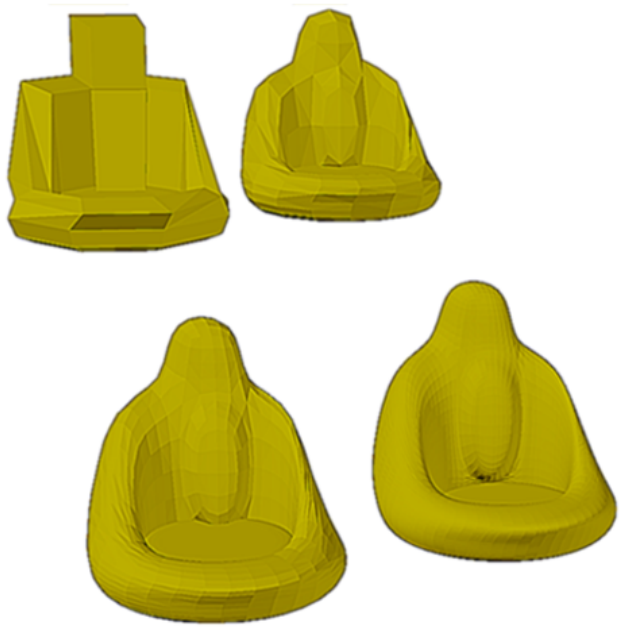
Increase the roundness of mesh objects by increasing the smoothness levels.

Mesh objects are made up of multiple subdivisions, or tessellations, which define the editable faces. Each face consists of underlying facets. When you increase smoothness, you increase the number of facets to provide a smoother, more rounded look.



Increase or Decrease Smoothness

As you work, you can increase and decrease the level of smoothness. The differences are apparent both in the wireframe and conceptual visual styles and in the rendered output.



The lowest level of smoothness, or baseline, is 0. By default, Level 0 has no smoothness. You can increase the smoothness of any mesh object up to the current limits. However, you cannot decrease the smoothness of a mesh object whose level of smoothness is zero.

If you have added creases to a mesh object, the effect of smoothing differs, depending on the crease setting. The effect of creases added to mesh that has no smoothness (Level 0) is not apparent until the mesh is smoothed.

As you edit an object using gizmos or grips, you might create gaps in the mesh object. One way to close the gap is to smooth the object or refine individual subobjects. Using hardware acceleration might also help resolve this problem. (For more information, see Graphics System.)

Limit Mesh Density

Mesh is created at the level of smoothness that you specify. The smoothness can range from None (0), to the default maximum (6), or to a level that you specify. As an object is smoothed, the density of the mesh facet grid also increases. For best results, model mesh objects at lower smoothness levels and increase the smoothness only after modeling is complete.

Dense meshes can result in subobjects that are difficult to select and edit. They can also affect performance. Therefore you might want to set limits that prevent the mesh from becoming too dense.

- **Maximum level of smoothness at which a grid is displayed** (SMOOTHMESHGRID). Displays the effects of modeling without the complexity of the underlying facet grid. The default smoothness level is 3. The tessellation display becomes increasingly dense until the maximum level is exceeded. Beyond that level, the display reverts to the most basic level, even though the smoothing level can continue to increase.
- **Maximum number of faces in a drawing** (SMOOTHMESHMAXFACE). Sets the maximum number of mesh faces that are permitted per mesh object.
- **Maximum level of smoothness** (SMOOTHMESHMAXLEV). Sets the maximum smoothness level permitted for mesh objects.

To increase the smoothness of a mesh object

1 Click Mesh Modeling tab ➤ Mesh panel ➤ Smooth More.

2 Select the mesh objects that you want to modify.

The smoothness of each object that you select is increased one level.

Toolbar: Smooth Mesh
Command entry: MESHSMOOTHMORE

To decrease the smoothness of a mesh object

1 Click Mesh Modeling tab ➤ Mesh panel ➤ Smooth Less.

2 Select the mesh objects that you want to modify.

The smoothness of each object that you select is decreased one level.

Toolbar: Smooth Mesh
Command entry: MESHSMOOTHLESS

To increase or decrease the smoothness of a mesh object (Properties palette)

1 If the Properties palette is not displayed, select any object. Right-click the object to display the shortcut menu. Click Properties.

2 Select the mesh objects that you want to modify.

3 In the Properties palette, Geometry area, Smoothness box, select a new smoothness level.

Toolbar: Standard 
Command entry: PROPERTIES

To control the display of the mesh facet grid

1 At the Command prompt, enter **smoothmeshgrid**.

2 Enter one of the following values:

- **0** suppresses the display of the underlying mesh facet grid.
- **1** displays the mesh facet grid only when the object has a level of smoothness of 0 or 1.
- **2 and higher** sets the highest level of smoothness at which the facet grid is displayed.

To set the maximum smoothness level for mesh objects

1 At the Command prompt, enter **smoothmeshmaxlev**.

2 Enter a value that represents the highest level of smoothness permissible for mesh objects. The recommended range is 1 - 5.

To set the maximum number of mesh faces

1 At the Command prompt, enter **smoothmeshmaxface**.

2 Enter the maximum number of faces permitted for an object.

You can set a value up to 16,000,000.

Quick Reference

Commands

MESHSMOOTHLESS

Decreases the level of smoothness for mesh objects by one level.

MESHSMOOTHMORE

Increases the level of smoothness for mesh objects by one level.

PROPERTIES

Controls properties of existing objects.

System Variables

SMOOTHMESHGRID

Sets the maximum level of smoothness at which the underlying mesh facet grid is displayed on 3D mesh objects.

SMOOTHMESHMAXFACE

Sets the maximum number of faces permitted for mesh objects.

SMOOTHMESHMAXLEV

Sets the maximum smoothness level for mesh objects.

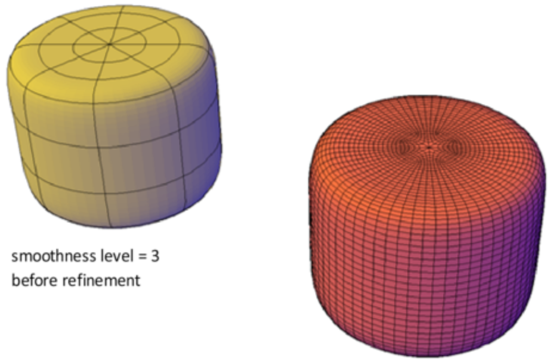
Refine Mesh Objects or Subobjects

Refine a mesh object or subobject to convert underlying facets to editable faces.

You can refine any mesh that has a level of smoothness of 1 or higher.

Refine a Mesh Object and Reset the Baseline

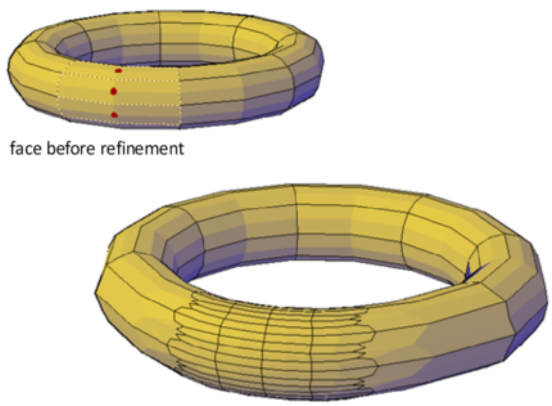
Refining an object increases the number of editable faces by converting the underlying facets to faces. The number of resulting faces depends on the current level of smoothness. Higher smoothness levels result in a higher number of faces after refinement.



In addition to increasing the number of faces, refining a mesh object resets its level of smoothness back to the baseline. Therefore, an object might appear to be smoothed, but its smoothness level can still equal 0 (zero).

Refine a Mesh Face

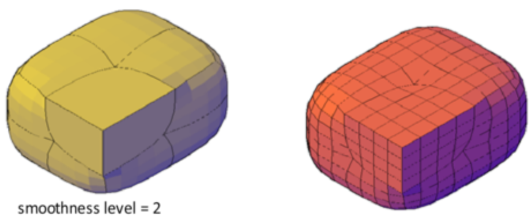
You can refine an entire mesh object as shown in the previous illustration, or select a specific face to refine. A refined face is subdivided into four faces and the surrounding faces are deformed slightly to accommodate the change.



Refining a mesh face does not affect the overall smoothing level of the mesh object. Unlike a refined mesh object, refined faces can be refined again immediately. With mesh face refinement, you can target smaller areas for detailed modeling.

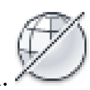
How Refinement Affects Creases

A crease that is set to Always retains its sharpness no matter how much you smooth or refine the object. However, the behavior is different when you assign a crease value. If you refine an object or edge that has a crease value, the assigned crease value is lowered by the value of the original level of smoothing. Suppose that you add a crease with a crease value of 4 and then refine a mesh whose level of smoothness is 2. The new crease value is 2.



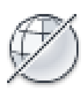
If a crease is applied before an object is smoothed or refined, the effect is not apparent until after the object is smoothed or refined.

To refine a mesh object

1 Click Mesh Modeling tab ➤ Mesh panel ➤ Refine Mesh. 

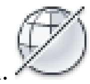
2 Select the mesh object that you want to refine. (The object must have a smoothness level of 1 or higher.)

  The underlying mesh facets are converted to faces and the level of smoothness for the object is set to 0. Higher levels of smoothness result in a higher number of faces.

Toolbar: Smooth Mesh 
Command entry: MESHREFINE

To refine a mesh face

1 Set the subobject selection filter to select only faces: right-click anywhere in the drawing area. Click Subobject Selection Filter ➤ Face.

2 Click Mesh Modeling tab ➤ Mesh panel ➤ Refine Mesh. 

3 Press Ctrl+click one or more mesh faces that you want to refine. Press Enter. (The object must have a smoothness level of 1 or higher.)

Each face is subdivided into four new faces.

Toolbar: Smooth Mesh 
Command entry: MESHREFINE

Quick Reference

Commands

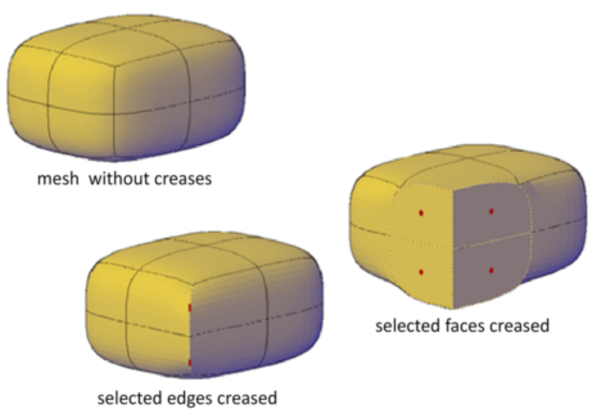
MESHREFINE

Multiplies the number of faces in selected mesh objects or faces.

Add Creases to Mesh

Add creases to sharpen mesh edges.

You can add creases to mesh objects that have a smoothing level of 1 or higher.



Add Creases to Different Subobjects

The result of creasing differs, depending on what type of subobject you select.

- **Edge**. The selected edge is sharpened. The adjacent faces are deformed to accommodate the new crease angle.
- **Face**. The selected face is flattened and all edges that bound that face are sharpened. Adjacent faces are deformed to accommodate the new shape of the face.
- **Vertex**. The point of the vertex and all intersecting edges are sharpened. Adjacent faces are deformed to accommodate the new vertex angle.

Assign a Crease Value to the Edge

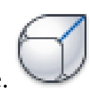
As you apply a crease, you set a crease value that determines how the crease is affected by smoothing. A value of Always ensures that the crease is always retained, even when the mesh is repeatedly smoothed. Higher crease values ensure that the crease is retained through several smoothing processes. (During smoothing, the assigned crease value is decreased by the value of the original level of smoothing.)

You can add a crease to mesh that has not been smoothed. However, the effect is not visible unless you smooth the object.

Remove a Crease

You can restore a crease to a smoothed state that corresponds to the smoothing level for the object. If you remove a crease that is adjacent to other creased subobjects, their contours are adjusted.

To add creases to a mesh object

1. (Optional) Specify the type of subobject to crease: right-click in the drawing area and click Subobject Selection Filter ➤ Face, Vertex, or Edge.
2. Click Mesh Modeling tab ➤ Mesh panel ➤ Add Crease. 
3. Select the mesh edges, faces, or vertices to crease. (If you have set a subobject selection filter, only one type of subobject can be selected.)

   To remove a subobject from the selection set, Shift+click the subobject.
4. Specify the crease value:
   - **Always** retains the crease at all levels of smoothness.

- **Values of 1 or higher** sets the level of smoothness that starts to affect the crease.

The specified subobjects are creased. A crease is not visible on objects that have not been smoothed (the level of smoothness = 0).

Toolbar: Smooth Mesh 
Command entry: MESHCREASE

To change the crease value of an existing mesh crease (Properties palette)

1 If the Properties palette is not displayed, select any object. Right-click the object to display the shortcut menu. Click Properties.

2 Press Ctrl+click the mesh subobject that you want to modify.

NOTE If you cannot select a specific subobject, verify whether subobject selection filtering is turned on for a different subobject type. (Right-click in the drawing area and click Subobject Selection Filter.)

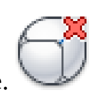
---

3 In the Properties palette, Crease area, Type box, change the crease value:
  - **None** removes the crease and sets the subobject to the current level of smoothness.
  - **Always** retains the crease at all levels of smoothness.
  - **By Level** sets the level of smoothness that starts to affect the crease. When this setting is selected, you can specify the crease level in the Level box.

Toolbar: Standard 
Command entry: PROPERTIES

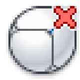
To remove an existing mesh crease

1 (Optional) Specify the type of subobject to modify: right-click in the drawing area and click Subobject Selection Filter ➤ Face, Vertex, or Edge.

2 Click Mesh Modeling tab ➤ Mesh panel ➤ Remove Crease. 

3 Press Ctrl+click the mesh subobjects to be modified and press Enter.

You can also use window selection to specify multiple subobjects.

Toolbar: Smooth Mesh 
Command entry: MESHUNCREASE

Quick Reference

Commands

MESHCREASE

Sharpens the edges of selected mesh subobjects.

MESHUNCREASE

Removes the crease from selected mesh faces, edges, or vertices.

PROPERTIES

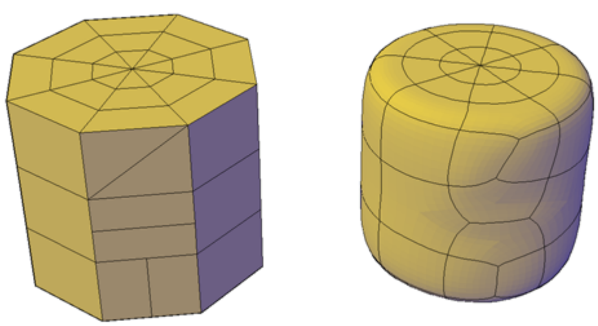
Controls properties of existing objects.

Split or Extrude Mesh Faces

Split a mesh face or deform it through extrusion.

Split a Mesh Face

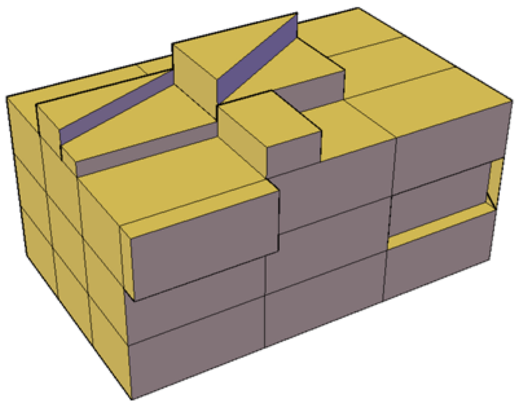
You can split a mesh face to make custom subdivisions. Use this method to prevent deforming a larger area for small modifications.



Because you specify the start point and end point of the split, this method also gives you control over the shape of the two new faces.

Extrude a Mesh Face

You can add definition to a 3D object by extruding a mesh face. Extruding other types of objects creates a separate 3D solid object. However extruding a mesh face extends, or deforms, the existing object and subdivides the extruded face.



You can use the same methods for extrusion of the faces of 3D solids and meshes as you use for other types of objects. For example, you can specify an extrusion direction, a path, or a taper angle. For more information about extrusion, see Extrude Objects on page 1199.

To split a mesh face

1. Click Mesh Modeling tab ➤ Mesh Edit panel ➤ Split Mesh Face. 
2. Press Ctrl+click the mesh face that you want to split.

   NOTE If you cannot select a face, verify whether subobject selection filtering is turned on for a different subobject type. (Right-click in the drawing area and click Subobject Selection Filter.)

   ---
3. Click the location on the edge where you want to start the split.
4. Click the location on the edge where you want to end the split.

   The mesh face is split along the boundary you specified. Surrounding faces are adjusted along the new faces.

Command entry: MESHSPLIT

To extrude a mesh face

1 Click Mesh Modeling tab ➤ Mesh Edit panel ➤ Extrude Face.

2 Press Ctrl+click a mesh face.

NOTE If you cannot select a face, verify whether subobject selection filtering is turned on for a different subobject type. (Right-click in the drawing area and click Subobject Selection Filter.)

3 Drag the face in the direction of the extrusion. Then specify the amount of extrusion using one of the following methods:

- Press Enter to set the extrusion dynamically.
- Enter a value to indicate the height or depth of the extrusion.

The selected face is subdivided and extruded. The new, subdivided faces are created around its boundaries to connect the extrusion with the original adjacent faces.

Command entry: EXTRUDE

Quick Reference

Commands

MESHSPLIT

Splits a mesh face into two faces.

EXTRUDE

Extends the dimensions of a 2D object or 3D face into 3D space.

System Variables

DELOBJ

Controls whether geometry used to create 3D objects is retained or deleted.

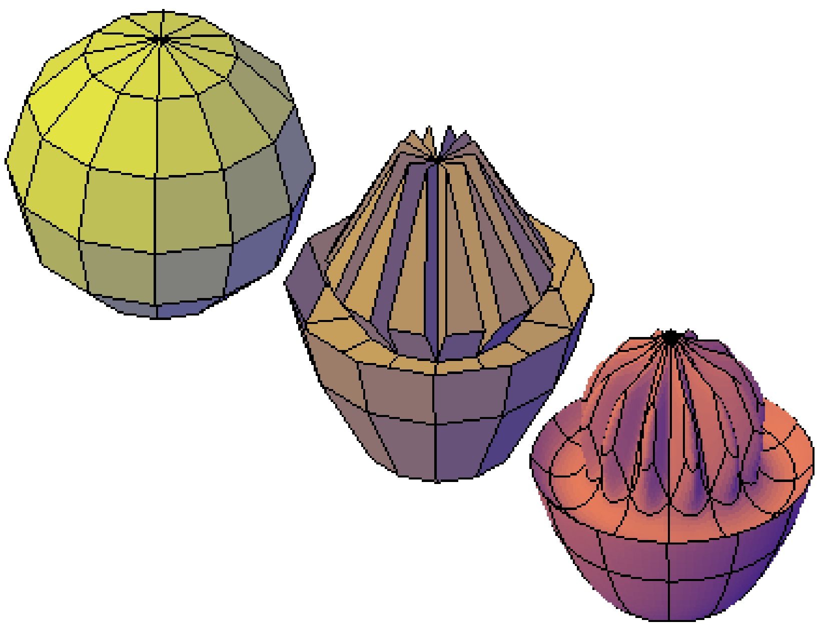
Tips for Working with Mesh

Learn some best practices for working with mesh models.

Mesh, with its enhanced modeling capabilities, offers a way to create more fluid, free-form designs. Keep these tips in mind as you work.

Model mesh before you smooth it.

Mesh modeling is a powerful way to design, but higher levels of smoothness increase complexity and can affect performance. You can work more efficiently if you complete editing operations such as gizmo editing, extrusion, and face splitting, on mesh objects that have not been smoothed. (That is, their level of smoothness is 0.)

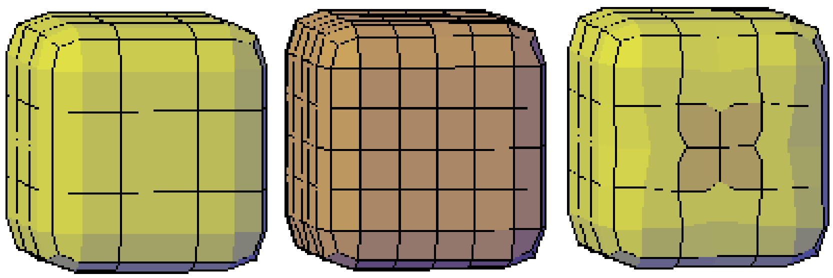


mesh sphere modeled by grip editing and extrusion, then smoothed

You can quickly switch between the levels of smoothness in the Properties palette to get a preview of how your activities affect the smoothed object.

Refine or split a face instead of refining the entire object.

Refinement is a powerful way to subdivide faces. However, by increasing the number of faces, you add to the overall complexity of the model. In addition, refining an entire mesh object resets the base level of smoothness to 0. This change can result in a dense grid that can no longer be simplified. For best results, avoid refining the object, and refine or split only the individual faces that require more detailed modeling.

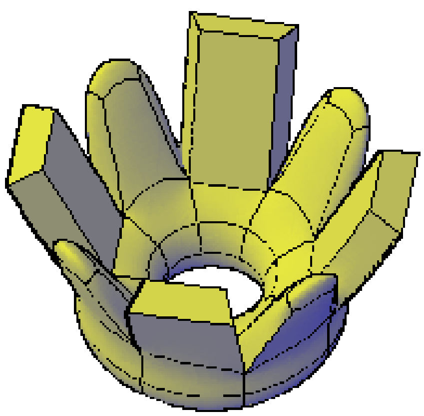


mesh box, refined mesh box, and mesh box with one face refined

Refining individual faces does not reset the level of smoothness for the object.

Crease edges to help limit distortion when the object is smoothed.

Creased edges can be set to maintain their sharpness, no matter how much the object is smoothed. You may also need to crease the edges in surrounding faces to obtain the result you want.

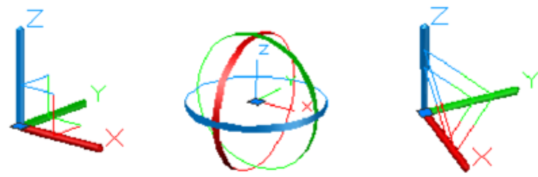


extruded faces on mesh torus, creased and not creased

Creasing set to Always retains its sharpness after smoothing. If you set a crease value, the creased edge becomes smoother at the equivalent level of smoothness.

Use gizmos to model faces, edges, and vertices.

3D Move, 3D Rotate, and 3D Scale gizmos can be used to modify entire mesh objects, or specific subobjects.



3D Move gizmo   3D Rotate gizmo   3D Scale gizmo

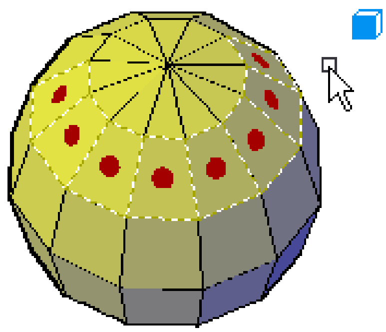
For example, you can rotate and scale an individual face using the 3D Move, Rotate, and Scale gizmos.

By constraining the modifications to a specified axis or plane, gizmos help you avoid unexpected results. The default gizmo is displayed whenever you select an object in a view that uses a 3D visual style. (You can also suppress this display.) Therefore, you do not have to explicitly start the 3D Move, 3D Rotate, or 3D Scale command to initiate these activities. You just need to select an object.

When a gizmo is selected, you can use the shortcut menu to switch to a different type of gizmo.

Use subobject selection filters to narrow the available selection candidates.

In a smoothed mesh, trying to select a specific subject can be difficult unless you turn on subobject selection (shortcut menu). By specifying that the selection set is limited to faces, edges, vertices, or even solid history subobjects, you can restrict which subobject type is available when you press Ctrl and move the mouse over the object.

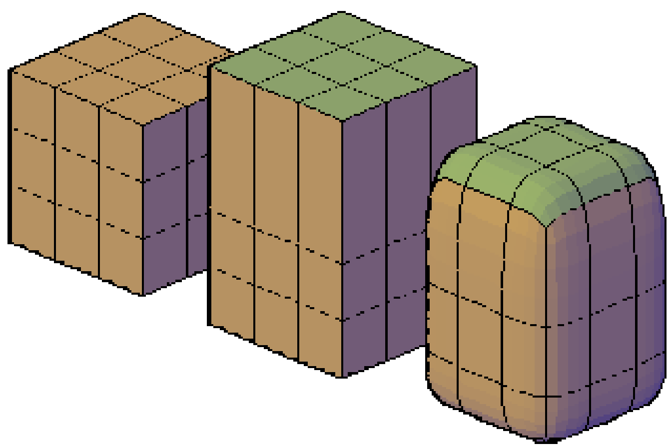


mesh faces selected when the face subobject selection filter is on

A filter is especially valuable for selecting mesh vertices, which are not highlighted as you move the mouse over them.

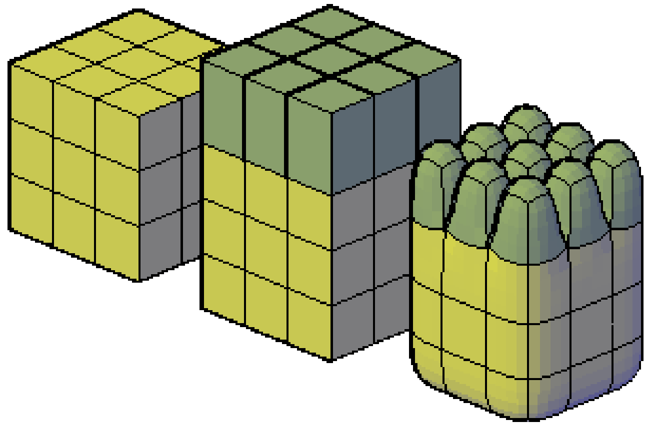
Model by extruding faces.

A key difference between gizmo editing and extrusion occurs in the way each face is modified. With gizmo editing, if you select and drag a set of faces, adjacent faces are stretched to accommodate the modification. When the object is smoothed, the adjacent faces adapt to the new location of the face.



mesh faces extended using 3D Move gizmo, then smoothed

Mesh extrusion, however, inserts additional faces to close the gap between the extruded face and its original surface.



mesh faces extruded, then smoothed

If you are working on an object that has not been smoothed, try smoothing it periodically to see how it is affected by smoothing.

Convert between mesh and 3D solids or surfaces.

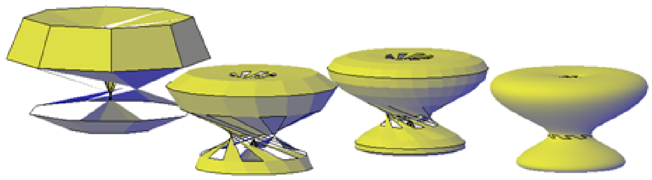
Mesh modeling is powerful, but it cannot do everything that solid modeling can do. If you need to edit mesh objects through intersection, subtraction, or union, you can convert mesh to 3D solid or surface objects. Similarly, if you need to apply creasing or smoothing to 3D solid or surface objects, you can convert those objects to mesh.

Keep in mind that not all conversions retain complete fidelity to the shape of the original object. Avoid switching between object types more than once, if possible. If you notice that the conversion modifies the shape of the object in an unacceptable way, undo the conversion and try again with different settings.

- The Mesh Tessellation Options Dialog Box (MESHOPTIONS) controls the smoothness and shape of the faces for 3D solids or surfaces that are converted to mesh. Although you can convert an object to mesh without opening this dialog box (MESHSMOOTH), you can more easily experiment with different conversion settings by launching the conversion operation from within the dialog box.
- The SMOOTHMESHCONVERT system variable (also available on the ribbon) sets whether the mesh objects that you convert to 3D solids or surfaces are smoothed or faceted, and whether their co-planar faces are optimized (merged).

You might have trouble converting some non-primitive mesh to solid objects due to the following problems:

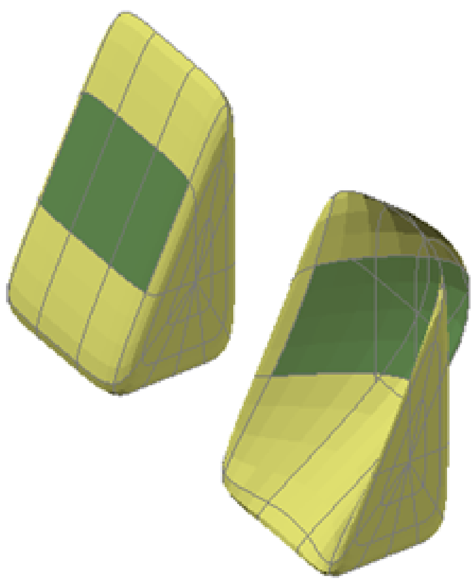
- **Gaps in the mesh.** If you notice gaps, you can sometimes close them by smoothing the object or by refining the faces that are adjacent to the gap.



mesh torus that has been twisted using 3D Rotate at various smoothing levels

In some cases, you can also obtain better results by using hardware acceleration to improve your graphics system.

- **Intersecting mesh faces.** Be especially careful not to create *self-intersections* as you move, rotate, or scale subobjects. (You create self-intersections when you cause one or more faces to cross, or intersect other faces in the same mesh model.) View the object from all viewpoints to ensure you create a viable model.



mesh wedge with front faces dragged past the back faces

Mesh objects that cannot be converted to solids can often be converted to surfaces instead.

Create Sections and 2D Drawings from 3D Models

22

Create cross sections, cutting planes, and flattened views of 3D objects.

Work with Sections

Create cross sections of 3D models.

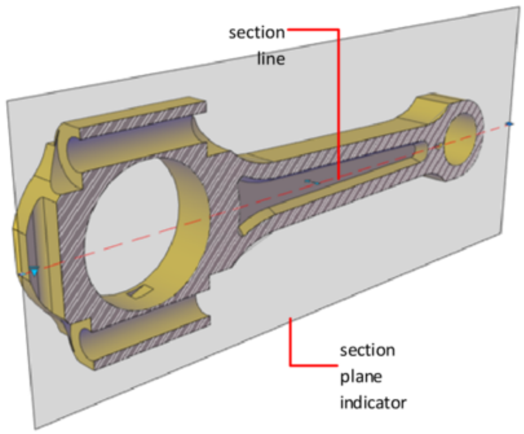
Overview of Section Objects

Create a section plane that can be modified and moved to achieve the cross section view that you need.

With the SECTIONPLANE command, you can create one or more section objects and place them throughout a 3D model (3D solids, surfaces, or mesh). By activating live sectioning, you can then view transient cuts in the 3D model as you move the section object through it. The 3D objects themselves do not change.

Set the Cross-Section with the Section Plane Indicator

Section objects have a transparent section plane indicator that acts as a cutting plane. This plane can be moved through a 3D model that is composed of 3D solids, surfaces, or regions to obtain different section views.

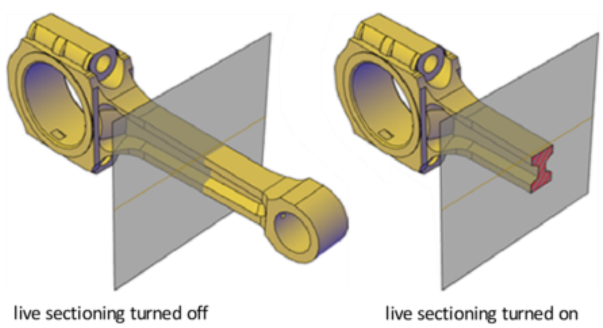


Store Properties in Section Lines

The section plane contains a *section line* that stores section object properties. You can create multiple section objects to store different properties. For example, one section object can display a hatch pattern at the section plane intersection. Another section object can display a specific linetype for the boundary of the intersected area.

Analyze the Model with Live Sectioning

With live sectioning, you can dynamically analyze the interior details of 3D objects by moving and adjusting the section plane. You can specify whether to hide, or cut away, the portion of the model that is on the viewing side of the section plane indicator.



Save and Share Section Images

After you create a sectional view, you can generate an accurate 2D or 3D block from the 3D model. These blocks can be analyzed or checked for clearances and interference conditions. They can also be dimensioned, or used as wireframe or rendered illustrations in documentation and presentation drawings.

You can also save each section object as a tool on the tool palette. That way, you can avoid resetting properties each time you create a section object.

Quick Reference

Commands

LIVESECTION

Turns on live sectioning for a selected section object.

SECTIONPLANE

Creates a section object that acts as a cutting plane through 3D objects.

SECTIONPLANESETTINGS

Sets display options for the selected section plane.

SECTIONPLANETOBLOCK

Saves selected section planes as 2D or 3D blocks.

Create Section Objects

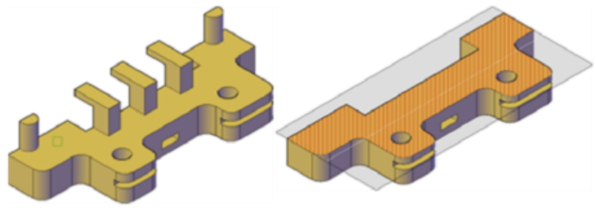
Create cross sections to show interior details of 3D objects.

With the SECTIONPLANE command, you create a *section object* that acts as a cutting plane through solids, surfaces, meshes, or regions. Then turn on *live sectioning* to move the section object through the 3D model to reveal its inner details in real time.

You can align a section object using several methods.

Align the Section Plane to a 3D Face

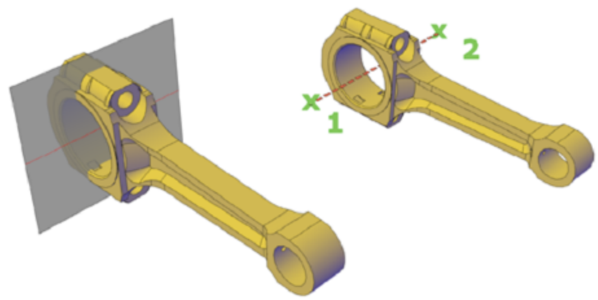
One way to set the section plane is to click the face of an existing 3D object. (As you move the cursor, a dotted outline indicates the side of the plane to be selected.) The section plane is automatically aligned to the plane of the face you select.



Section object aligned to face

Create a Straight Cutting Plane

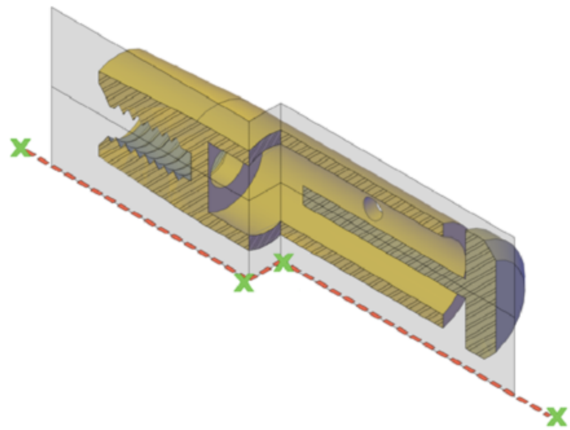
Pick two points to create a straight cutting plane.



Add a Jogged Segment

The section plane can be a straight line or it can have multiple or jogged sections. For example, a section containing a jog is one that cuts away a pie slice-shaped wedge from a cylinder.

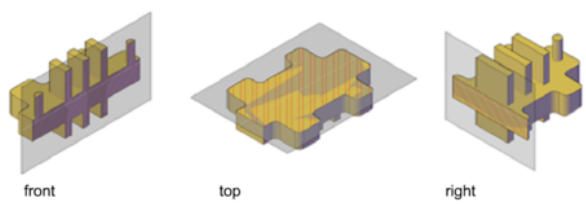
Create a section line that has jogged segments by using the Draw Section option to pick multiple points throughout the 3D model.



Section object with jogged segment

Create Orthographic Sections

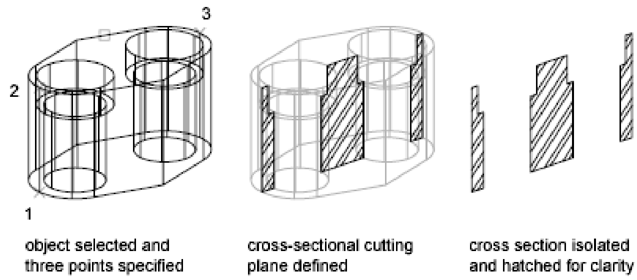
You can align section objects to a specified orthographic orientation of the current UCS, such as front, back, bottom, top, left, or right.



Orthographic section planes are placed so that they pass through the center of the 3D extents of all 3D objects in the drawing.

Create a Region to Represent the Cross Section

With the SECTION command, you can create a 2D region object that represents a planar cross section through a 3D solid object. You do not have live sectioning capabilities when you use this legacy method to create cross sections.



Define the plane of the cross section using one of the following methods:

- Specify three points.
- Specify a 2D object such as a circle, ellipse, arc, spline, or polyline.
- Specify a view.
- Specify the *Z* axis.
- Specify the *XY*, *YZ*, or *ZX* plane.

The new region that represents the cross-sectional plane is placed on the current layer.

NOTE Before you apply hatching to the cross-sectional cutting plane, align the UCS with the cutting plane.

To create a section object by selecting a face

1. Click Home tab ➤ Section panel ➤ Section Plane.
2. Click to select a face on your model.
   A section object is created on the plane of the selected face.
3. Click the section line to display its grips.
4. Select a grip to move the section plane through the 3D object.
   A section object is created in the Section Plane state. Live sectioning is turned on.

Command entry: SECTIONPLANE

To create a section object by specifying two points

1 Click Home tab ➤ Section panel ➤ Section Plane.

2 Specify the first point of the section object.

3 Specify the endpoint.

The section object is created between the two points. Live sectioning is turned off.

Command entry: SECTIONPLANE

To create a section object with jogged segments

1 Click Home tab ➤ Section panel ➤ Section Plane.

2 At the Command prompt, enter **d** (Draw Section).

3 Specify the start point of the section object.

4 Specify a second point to create the first jogged segment.
From this point, you cannot create segments that intersect.

5 Continue specifying segment endpoints, then press Enter.

6 Specify a point in the direction of the sectional cut.

A section object with multiple segments is created in a Section Boundary state. Live sectioning is turned off.

Command entry: SECTIONPLANE

To create a section object on a preset orthographic plane

1 Click Home tab ➤ Section panel ➤ Section Plane.

2 At the Command prompt, enter **o** (Orthographic).

3 Select an alignment option.

The new section object intersects the center point of the 3D extents of all 3D objects in the drawing. It is placed on the selected orthographic plane. Live sectioning is turned on.

Command entry: SECTIONPLANE

To create a region that represents the cross section of a 3D solid object

1. At the Command prompt, enter **section**.
2. Select the object to cross section.
3. Specify three points to define the cross-sectional plane.

Command entry: SECTION

Quick Reference

Commands

SECTIONPLANE

Creates a section object that acts as a cutting plane through 3D objects.

SECTION

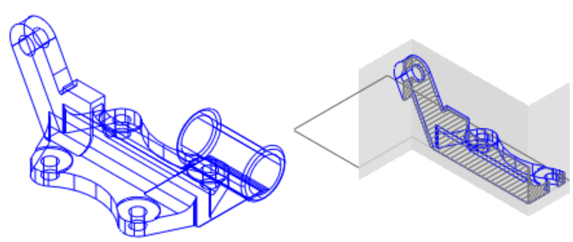
Uses the intersection of a plane and solids, surfaces, or mesh to create a region.

Modify a Section View

After you create a section, adjust its display or modify its shape and location to change the represented section view.

Add Jogs to a Section

Add jogs, or angular segments, to existing section lines.



You can create a section line that has multiple segments (jogs), using the Draw Section option of the SECTIONPLANE command. You can also add a jog to an existing section object using the Add Jog to Section option on the shortcut menu with SECTIONPLANEJOG.

A jog that is added to an existing section object creates a segment that is perpendicular to the selected segment. Its viewpoint is oriented in the direction set by the Direction grip. The Nearest object snap is temporarily turned on to help you place the jogs on a section.

You cannot add jogs to the side or back lines of the section object.

After adding jogs, you can reposition and resize the jogged sections by dragging the section object grips.

To add jogs to a section

1. Click Home tab ➤ Section panel ➤ Jog.
2. On a section object, select the section line.
3. Move the cursor over the section line.
4. Select a point on the section line where you want to place a jog that is perpendicular to the selected segment.

   To create additional jogs, repeat the steps.

NOTE You cannot create jogs that cause the line to intersect itself or close.

Shortcut menu: Add Jog to Section
Command entry: SECTIONPLANEJOG

Quick Reference

Commands

SECTIONPLANE

Creates a section object that acts as a cutting plane through 3D objects.

SECTIONPLANEJOG

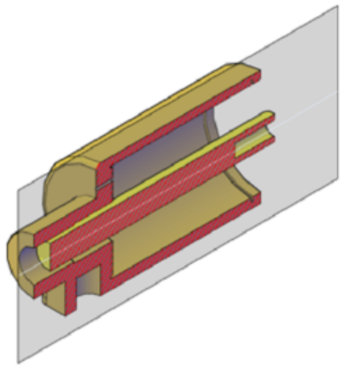
Adds a jogged segment to a section object.

Use Live Section to Adjust the Cross Section

Use live sectioning to move a section object through the 3D model or region dynamically.

What Is Live Sectioning?

Live sectioning is an analytical tool for viewing cut geometry in a 3D solid, surface, or region.

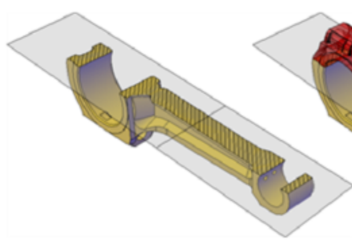


You can use live sectioning to analyze a model by moving the section object through the object. For example, sliding the section object through an engine assembly helps you visualize its internal components. You can use this method to create a cross section view that you can save or reuse.

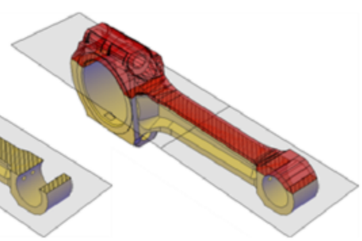
Turn on and Use Live Sectioning

Live sectioning works with 3D objects and regions in model space. When live sectioning is activated, you can change the viewing planes by using grips to adjust the location of the section object or its segments.

By turning on cutaway geometry, you can display the entire object that contains the section plane. This option (available on the shortcut menu) can only be turned on when section plane is active.



cut-away geometry not displayed



cut-away geometry displayed

Live sectioning is turned on or off automatically, depending on how you create the section object. For example, when you select a face to define the section plane, live sectioning is turned on. When you create sections using the Draw Section option of the SECTIONPLANE command, live sectioning is turned off. Live sectioning can be manually turned on or off after a section object is created.

A drawing can contain multiple section objects. However, live sectioning can only be active for one section object at a time. Suppose that your model has two sections objects: *Section A* and *Section B*. If *Section A* has live sectioning turned on and you activate live sectioning for *Section B*, live sectioning for *Section A* is automatically turned off.

Turning off a section object layer does not turn off live sectioning. However, freezing the layer turns off live sectioning.

To turn live sectioning on and off

1 Select a section object.

2 Click Home tab ➤ Section panel ➤ Live Section.

Shortcut menu: Activate Live Sectioning
Command entry: LIVESECTION

To display cutaway geometry

1 Select a section object.

2 Right-click the section line. Click Show Cut-away Geometry to turn it on.

   The cut geometry is displayed according to the Foreground Line settings in the Section Settings dialog box.

Quick Reference

Commands

LIVESECTION

Turns on live sectioning for a selected section object.

SECTIONPLANE

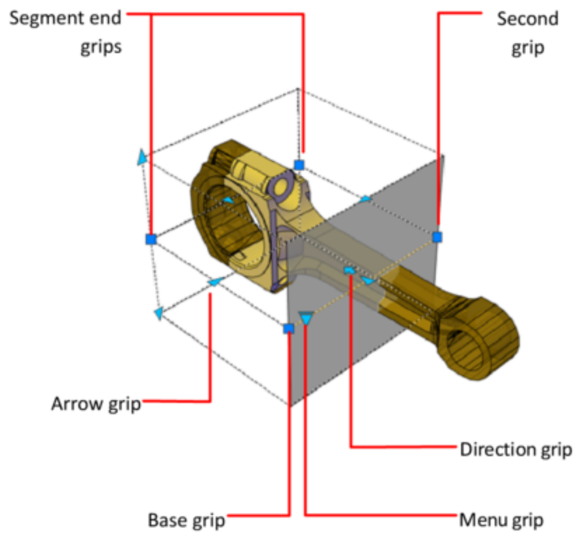
Creates a section object that acts as a cutting plane through 3D objects.

SECTIONPLANESETTINGS

Sets display options for the selected section plane.

Use Grips to Modify Section Objects

Section object grips help you move and resize the section object.



Grips allow you to adjust the location, length, width, and height of the cutting area.

- **Base grip**. Acts as the basepoint for moving, scaling, and rotating the section object. It is always adjacent to the Menu grip.
- **Second grip**. Rotates the section object around the base grip.
- **Menu grip**. Displays a menu of section object states, which control the display of visual information about the cutting plane.
- **Direction grip**. Controls the viewing direction of the 2D section. To reverse the viewing direction of the section plane, click the Direction grip.
- **Arrow grip**. (Section Boundary and Volume states only.) Modifies the section object by modifying the shape and position of the section plane. Only orthogonal movements in the direction of the arrow are permitted.
- **Segment end grips**. (Section Boundary and Volume states only.) Stretches the vertices of the section plane. You cannot move segment end grips so that segments intersect. Segment end grips are displayed at the endpoints of jogged segments.

You can select only one section object grip at a time.

To adjust a section object using grips

1. In the drawing area, click the section plane.

   Depending on the current section plane state, different grips are displayed.
2. Hover over the grip with the cursor until it turns red. Then drag the grip to a new location.
3. To adjust the height, width, or depth of a section plane, click the Menu grip and select Section Volume or Section Boundary.

Quick Reference

Commands

SECTIONPLANE

Creates a section object that acts as a cutting plane through 3D objects.

System Variables

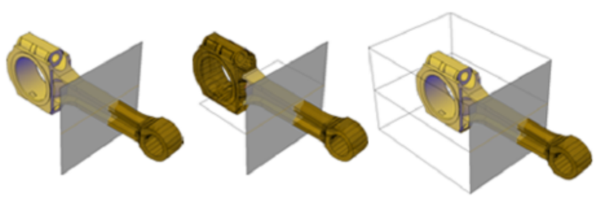
GRIPSIZE

Sets the size of the grip box in pixels.

Set Section Object States and Properties

Set the display of the section object.

Set Section Object States



section plane state | section boundary state | section volume state

Section objects have the following display states:

- **Section Plane.** The section line and transparent section plane indicator are displayed. The cutting plane extends infinitely in all directions.

- **Section Boundary.** A 2D box shows the *XY* extents of the cutting plane. The cutting plane along the *Z* axis extends infinitely.
- **Section Volume.** A 3D box shows the extents of the cutting plane in all directions.

You can switch between object states by clicking the Menu grip that is displayed when you select the section object.

Set Section Object Properties

Section objects have properties like other AutoCAD objects. Properties are stored in the section line and can be accessed in the Properties palette.

For each section object, you can change the name, layer, and linetype. You can also change the color and transparency of the section plane indicator (the cutting plane).

To change the state of a section object by using the Menu grip

1. Select the section object to display its grips.
2. Click the Menu grip.
3. On the section state menu, click the state you want to display.
   The section display is updated to the selected state.

To change the state of a section object (Properties palette)

1. Right-click the section object. Click Properties.
2. In the Properties palette, under Section Object, change the value for Type.
   The section display is updated to the selected state.

Command entry: PROPERTIES

To rename a section object

1. Right-click the section object. Click Properties.
2. In the Properties palette, under Section Object, in the Name box, enter a different name.

Command entry: PROPERTIES

To change the transparency and color of the section plane indicator

1. Right-click the section object. Click Properties.
2. In the Properties palette, under Section Object, in the Plane Transparency box, enter a value of 1-100. Entering 1 makes the section plane indicator opaque.
3. In the Plane Color box, select a color.

   The section plane indicator is updated in the viewport.

Command entry: PROPERTIES

To change the height of the section plane indicator

1. Right-click the section object. Click Properties.
2. In the Properties palette, under Geometry, in the Top Plane box, change the value.

   This value changes the distance from the section line to the upper edge of the section plane.
3. In the Bottom Plane box, enter a value.

   This value changes the distance from the section line to the bottom edge of the section plane.

   The section plane indicator is updated in the viewport.

Command entry: PROPERTIES

To modify the live section display settings

1. Click Home tab ➤ Section panel ➤ Panel dialog box launcher.
2. Right-click the section object. Click Live Section Settings.
3. In the Section Settings Dialog Box, click Live Section Settings.
4. Modify the sections that you want to change and click OK.

Shortcut menu: Live Section Settings
Command entry: SECTIONPLANESETTINGS

Quick Reference

Commands

PROPERTIES

Controls properties of existing objects.

SECTIONPLANE

Creates a section object that acts as a cutting plane through 3D objects.

SECTIONPLANESETTINGS

Sets display options for the selected section plane.

SECTIONPLANETOBLOCK

Saves selected section planes as 2D or 3D blocks.

Associate Section Objects with Views and Cameras

Associate section objects with named views and cameras in the View Manager.

When you activate a named view or camera that has an associated section object, live sectioning is turned on for that section object. For a 3D model with multiple section objects, you might want to associate a particular section object to a view or camera. Later, you can restore a saved sectional view or camera and activate live sectioning for the associated section object.

For example, you can set up two section objects that cut through the 3D model in different directions. *Section object A* cuts the model along its width; *Section object B* cuts the model along its length. Perhaps you want to view the sectional cut that is perpendicular to your line of sight. By associating each section object with a view or camera, you can quickly switch between the two views and see the desired cross section.

To associate a section object with a view or camera

1. Click View tab ➤ Views panel ➤ Named Views.
2. In the View Manager, in the left pane, under Model Views, select a named view or camera.
3. In the General section, in the Live Section drop-down list, select a section object by name.
4. Click OK.

The section object is now associated with a named view or camera. When the named view or camera is restored, live sectioning is turned on for the associated section object.

Toolbar: View 
Command entry: VIEW

Quick Reference

Commands

SECTIONPLANE

Creates a section object that acts as a cutting plane through 3D objects.

VIEW

Saves and restores named views, camera views, layout views, and preset views.

Save and Publish Section Objects

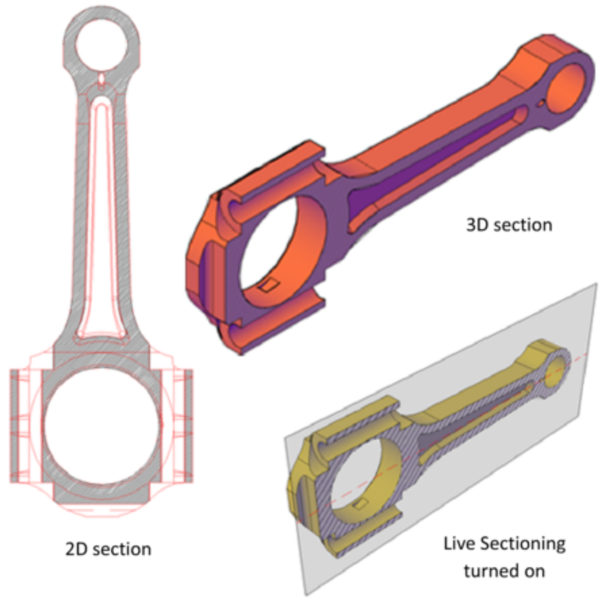
Save a section object as a block or tool, or publish it.

Save Sections as Blocks, Drawings, or Tools

Save the representation of the cross-sectional area where a section object intersects a 3D model as a block.

Save Sections as Blocks or Drawings

You can save the section objects you create as blocks. Working from the Generate Section / Elevation Dialog Box, you can choose the type of block that is created.



For example, suppose your project requires 2D elevation drawings or 2D cross sections. The 2D Section / Elevation option creates an accurate block representation that is ready for dimensioning.

To publish or render a cutaway of the 3D model, select the 3D Section option. 3D section geometry consists of mostly 3D solids and surfaces. However, profile outlines and hatch patterns consist of 2D lines.

The display properties of 2D section/elevation blocks and 3D section blocks are controlled in the Section Settings dialog box.

When you create section blocks, you have the following choices for how they are handled:

- **Insert the section blocks.** At the time of creation, you can insert a 2D or 3D section block into the drawing or save it to an external file. A 2D section block is inserted on the *XY* plane of the current UCS, including section blocks that extend into 3D space.
  Inserted section blocks are initially unnamed. You can set the scale, rotation, and basepoint upon insertion. You can modify and rename them later by editing the block with BEDIT.
- **Export section blocks to a file.** Save and name the new section objects so they can be inserted later.

- **Save section block components on separate layers.** By default, section block components such as intersection boundary, intersection fill, background lines, cutaway geometry, and curve tangency lines are saved on Layer 0. However, you can separate the components of saved section blocks onto separate layers with a suffix or prefix that you specify. Assigning a suffix or prefix helps you organize the block components into layers that you can sort and identify quickly. The Layer properties lists in the Section Settings dialog box provide the opportunity to customize the layer names.
- **Specify whether to limit the section block to certain objects.** The objects that are included in a section block vary, depending on which section object state is selected. You can also select specific objects to be included as you create the section block.

Save Section Objects as Tools

As with other objects, you can create tools from individual section objects and then access them in a tool palette when you need them.

If each of several section objects has its own set of properties, you can save each section object as a tool. Later, you can quickly create new section objects that use the same settings.

See also:

- Create and Use Tools from Objects and Images on page 56

To save and insert a 2D or 3D section as a block

1. Click Home tab ➤ Section panel ➤ Create Block. 
2. Select the section object.
3. In the Generate Section/Elevation dialog box, click 2D Section/Elevation or 3D Section.
4. In the expanded dialog box, select Include All Objects.
5. Under Destination, click Insert as New Block.
6. Click Create.
7. In the drawing area, specify an insertion point for the new block.

   An unnamed block is inserted consisting of 2D or 3D geometry.

Shortcut menu: Generate 2D/3D Section
Command entry: SECTIONPLANETOBLOCK

To save section block components on separate layers

1. Click Home tab ➤ Section panel ➤ Create Block.
2. Click a section object.
3. In the Generate Section/Elevation dialog box, click 2D Section/Elevation or 3D Section.
4. In the expanded dialog box, click Section Settings.
5. In the Section Settings dialog box, in the list of properties, click the Layer box under the section component that you want to update.
6. In the expanded Layer list, specify the layer on which to place the section block component:
   - To set an existing layer name, click *ObjectByLayer* or the name of another layer. Go to step 7.
   - To create a descriptive label to be added to the block section layer names, click New Layer Name Settings.
7. In the New Layer Name dialog box, specify how the descriptive text will look:
   - Under Added Text Type, specify whether you want the descriptive text to be a prefix or a suffix.
   - Under Text Added to the Existing Layer Name, enter descriptive text to be added to the layer name.

   Click OK.
8. In the Section Settings dialog box, click OK.
9. In the Generate Section / Elevation dialog box, click Create.

Shortcut menu: Generate 2D/3D Section
Command entry: SECTIONPLANETOBLOCK

To save a section object tool as a tool palette tool

1. On a section object, select the section line.

2 Drag the section object to the position on the tool palette where you want to place the tool.

   The black line indicates the new location of the tool.

3 Release the mouse button.

   The section object is saved as a tool on the tool palette.

Quick Reference

Commands

BEDIT

Opens the block definition in the Block Editor.

SECTIONPLANE

Creates a section object that acts as a cutting plane through 3D objects.

SECTIONPLANESETTINGS

Sets display options for the selected section plane.

SECTIONPLANETOBLOCK

Saves selected section planes as 2D or 3D blocks.

TOOLPALETTES

Opens the Tool Palettes window.

Publish Section Objects

Control the visibility of section objects when you render, plot, or view them in the DWF file viewer.

Render Section Objects

With live sectioning turned on, all lines on a section object are rendered as 2D lines. The section plane indicator is rendered as a transparent material. Its degree of transparency is controlled in the Properties palette.

If you want to render a 3D cutaway, save the cutaway section as a 3D block and render the block reference.

Plot Section Objects

When a section object is in a Section Boundary or Section Volume state, displayed lines cannot be plotted. The section plane indicator is plotted as if it were transparent. However, it does not have the same visual quality that it has when it is rendered.

If you do not want to plot the section line, place the section object on a layer that is turned off.

View Section Objects in the DWF file viewer

When live sectioning is active for a section object, the 3D model is displayed in the DWF Viewer with the best possible visual quality. Geometry that is hidden by live sectioning is also hidden in the DWF Viewer.

Display settings for live sections have the same appearance in the viewer as they have in the drawing. For example, dashed linetypes and hatch patterns assigned to intersected areas of a 3D model are retained in the DWF Viewer display.

The section object is not visible in the DWF Viewer.

Quick Reference

Commands

3DDWF

Creates a 3D DWF or 3D DWFx file of your 3D model and displays it in the DWF Viewer.

PLOT

Plots a drawing to a plotter, printer, or file.

PUBLISH

Publishes drawings to electronic sheet sets (DWF, DWFx, or PDF files) or plotters.

RENDER

Creates a photorealistic or realistically shaded image of a 3D solid or surface model.

SECTIONPLANE

Creates a section object that acts as a cutting plane through 3D objects.

SECTIONPLANESETTINGS

Sets display options for the selected section plane.

SECTIONPLANETOBLOCK

Saves selected section planes as 2D or 3D blocks.

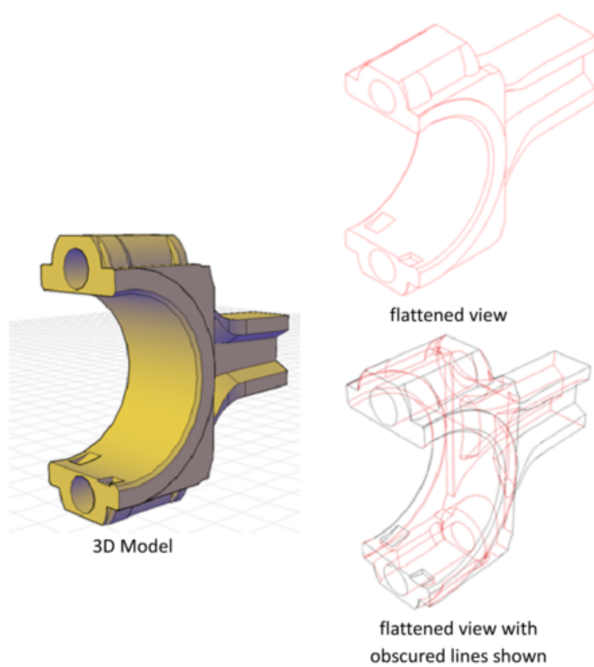
Create a Flattened View

Create a flattened view of the 3D solids and regions in the current view.

Create a Flatshot Object

With the FLATSHOT command, you can create a flattened, 2D representation of the 3D model projected onto the *XY* plane. The resulting objects can be inserted as a block or saved as a separate drawing.

The process is like taking a photograph of the entire 3D model and then laying the photograph flat. This feature is useful for creating technical illustrations.



The flatshot process works only in model space. Start by setting up the view you want, including orthographic or parallel views. All 3D objects in the model space viewport are captured. Therefore, be sure to place the objects you do not want captured on layers that are turned off or frozen.

As you create the block, you can control how hidden lines are displayed by adjusting the Foreground and Obscured Lines settings in the Flatshot dialog box. For best results with mesh objects, clear the Show box under Obscured Lines so that hidden lines are not represented.

Three-dimensional objects that have been sectioned are captured in their entirety, as if they had not been sectioned.

NOTE To create profile images of 3D solids in paper space, use the SOLPROF command.

Modify a Flatshot Block

You can modify a flattened view that has been inserted as a block in the same way that you modify any other 2D block geometry.

To create a flattened 2D view of a 3D model

1. Set up the view of the 3D model.
2. Click Home tab ➤ Section panel ➤ Flatshot.
3. In the Flatshot dialog box, under Destination, click one of the options.
4. Change the color and linetype settings for Foreground and Obscured lines.
5. Click Create.
6. Specify an insertion point on the screen to place the block. Adjust the basepoint, scale, and rotation if necessary.

   A block is created consisting of 2D geometry that is projected onto the *XY* plane of the current UCS.

Command entry: FLATSHOT

Quick Reference

Commands

FLATSHOT

Creates a 2D representation of all 3D objects based on the current view.

SOLPROF

Creates 2D profile images of 3D solids for display in a layout viewport.

Annotate Drawings

Work with Annotations 23

When you annotate your drawings, you can use certain tools and properties to make working with annotations easier.

Overview of Annotations

Annotations are notes or other types of explanatory symbols or objects that are commonly used to add information to your drawing.

Examples of annotations include

- Notes and labels
- Tables
- Dimensions and tolerances
- Hatches
- Callouts
- Blocks

The types of objects that you use to create annotations include

- Hatches
- Text (single-line and multiline)
- Tables
- Dimensions
- Tolerances
- Leaders and multileaders

- Blocks
- Attributes

Quick Reference

Commands

ATTDEF

Creates an attribute definition for storing data in a block.

BLOCK

Creates a block definition from selected objects.

DIMSTYLE

Creates and modifies dimension styles.

HATCH

Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

MLEADERSTYLE

Creates and modifies multileader styles.

MTEXT

Creates a multiline text object.

OBJECTSCALE

Adds or deletes supported scales for annotative objects.

STYLE

Creates, modifies, or specifies text styles.

TEXT

Creates a single-line text object.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Scale Annotations

You can automate the process of scaling annotations in various layout viewports and in model space.

Overview of Scaling Annotations

Objects that are commonly used to annotate drawings have a property called *Annotative*. This property allows you to automate the process of scaling annotations so that they plot or display at the correct size on the paper.

Instead of creating multiple annotations at different sizes and on separate layers, you can turn on the annotative property by object or by style, and set the annotation scale for layout or model viewports. The annotation scale controls the size of the annotative objects relative to the model geometry in the drawing.

The following objects are commonly used to annotate drawings and contain an annotative property:

- Text
- Dimensions
- Hatches
- Tolerances
- Multileaders
- Blocks
- Attributes

When the Annotative property for these objects is turned on (set to Yes), these objects are called *annotative objects*.

You define a paper size for annotative objects. The *annotation scale* you set for layout viewports and model space determines the size of the annotative objects in those spaces.

Save to Legacy Drawing File Format

Set the system variable SAVEFIDELITY to 1 when you save a drawing that contains annotative objects to a legacy drawing file format (AutoCAD 2007 or earlier). This preserves the visual fidelity of the drawing when it is opened in a release earlier than AutoCAD 2008 by saving individual representations of each scale of each annotative object. The individual objects are saved to layers that are used to organize objects of the same scale. Setting SAVEFIDELITY to 0, when opening the drawing in AutoCAD 2008 or later release, results in improved performance. For more information about saving a drawing to a previous release, see Save a Drawing on page 226.

Workflow for Annotating Drawings

The following steps represent a typical workflow for annotating a drawing so that your annotations will scale automatically.

1. Create annotative styles on page 1403.
2. In model space, set the annotation scale to the scale at which the annotations will be plotted or displayed on page 1396.
3. Create annotative objects using annotative styles on page 1403.

If one or more annotative objects needs to be displayed at an additional scale, follow these steps.

1. Add the additional scale to the annotative objects on page 1426.
2. Set the annotation scale to the new scale on page 1396 (the annotative objects that support the new scale will be resized based on the annotation scale).
3. Reposition the annotative objects as needed for the new scale

When you create your layouts, follow these steps.

1. Create a new layout on page 440 or make a layout current on page 442.
2. Create viewports on page 453.

3 Set the annotation scale for each viewport on page 1396. (For each viewport, the annotation scale and viewport scale should be the same).

For more information about setting visibility for annotative objects, see Display Annotative Objects on page 1422. For more information about adding scales to annotative objects, see Add and Modify Scale Representations on page 1424.

Quick Reference

Commands

OBJECTSCALE

Adds or deletes supported scales for annotative objects.

System Variables

ANNOAUTOSCALE

Updates annotative objects to support the annotation scale when the annotation scale is changed.

CANNOSCALE

Sets the name of the current annotation scale for the current space.

CANNOSCALEVALUE

Returns the value of the current annotation scale.

MSLTSCALE

Scales linetypes displayed on the model tab by the annotation scale.

Utilities

No entries

Command Modifiers

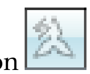
No entries

Set Annotation Scale

is a setting that is saved with model space, layout viewports, and model views. When you add objects to your drawing, they support the current annotation scale and are scaled based on that scale setting and automatically displayed at the correct size in model space.

Before you add annotative objects to your model, you set the annotation scale. Think about the eventual scale settings of the viewports in which the annotations will display. The annotation scale should be set to the same scale as the viewport in which the annotative objects will display in the layout (or the plot scale if plotting from model space). For example, if the annotative objects will display in a viewport that has a scale of 1:2, then you set the annotation scale to 1:2.

When working on the model tab or when a viewport is selected, the current annotation scale is displayed on the application or drawing status bar. You can use the status bars to change the annotation scale.

You can use the ANNOAUTOSCALE system variable to update annotative objects to support the current scale automatically when the annotation scale is changed. ANNOAUTOSCALE is turned off by default to keep file size down and improve performance. When ANNOAUTOSCALE is off, this button  is displayed this way on the right side of the drawing status bar or application status bar.

Use the CANNOSCALE system variable to set a default annotation scale setting.

See also:

■ Drawing Status Bar on page 37

To set the annotation scale while working on the Model tab

1 On the right side of the drawing or application status bar, click the arrow next to the displayed annotation scale.

2 Select a scale from the list.

To set the annotation scale for a layout viewport

1 On a layout tab, select a viewport.

2 On the right side of the drawing or application status bar, click the arrow next to the displayed annotation scale.

3 Select a scale from the list.

To set the default annotation scale

1. At the command prompt, enter **cannoscale**.
2. Enter a scale name. Press ENTER

Command entry: CANNOSCALE

Quick Reference

Commands

OBJECTSCALE

Adds or deletes supported scales for annotative objects.

System Variables

ANNOAUTOSCALE

Updates annotative objects to support the annotation scale when the annotation scale is changed.

CANNOSCALE

Sets the name of the current annotation scale for the current space.

CANNOSCALEVALUE

Returns the value of the current annotation scale.

MSLTSCALE

Scales linetypes displayed on the model tab by the annotation scale.

Utilities

No entries

Command Modifiers

No entries

Create Annotative Objects

Objects that are commonly used to annotate drawings have a property called *Annotative*. When the Annotative property for these objects is turned on (set to Yes), these objects are called *annotative objects*

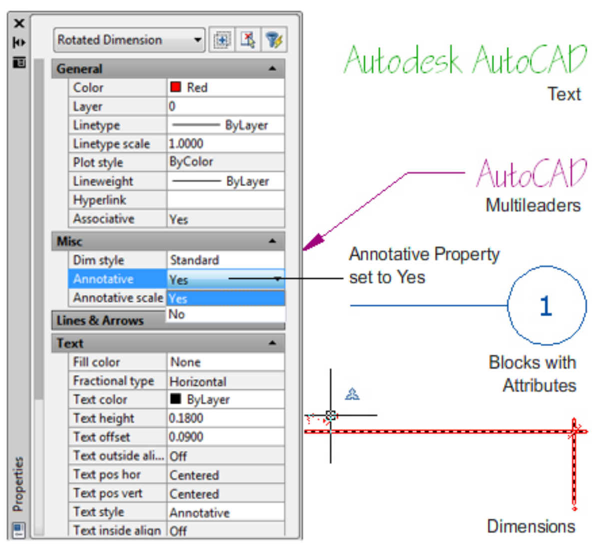
Overview of Creating Annotative Objects

When you add annotations to your drawing, you can turn on the property for those objects. These annotative objects are scaled based on the current setting and are automatically displayed at the correct size

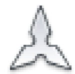
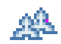
Annotative objects are defined at a paper height and display at the size determined by the annotation scale.

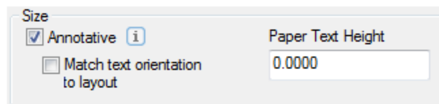
The following objects can be annotative (have an Annotative property):

- Hatches
- Text (single-line and multiline)
- Dimensions
- Tolerances
- Leaders and multileaders (created with MLEADER)
- Blocks
- Attributes

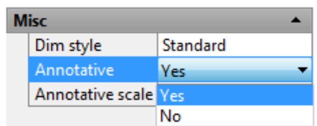


Many of the dialog boxes used to create these objects contain an Annotative check box where you can make the object annotative. You can also change existing objects to be annotative by changing the annotative property in the Properties palette.

When you hover the cursor over an annotative object that supports one annotation scale, the cursor displays a  icon. When the object supports more than one annotation scale, it displays a  icon.



From a dialog



From the Properties palette

Icon showing the Multileader is annotative

Text, dimension, and multileader styles can also be annotative. Annotative styles create annotative objects.

Visual Fidelity for Annotative Objects

When working with objects, this option allows you to maintain visual fidelity for these objects when they are viewed in AutoCAD 2007 and earlier releases. Visual fidelity is controlled by the SAVEFIDELITY system variable.

If you work primarily in model space, it is recommended that you turn off visual fidelity (set SAVEFIDELITY to 0). However, if you need to exchange drawings with other users, and layout fidelity is most important, then visual fidelity should be turned on (set SAVEFIDELITY to 1).

NOTE The SAVEFIDELITY system variable does not affect saving a drawing to the AutoCAD 2010 drawing or DXF file formats.

Annotative objects may have multiple . When visual fidelity is on, annotative objects are decomposed and scale representations are saved (in an) to separate layers, which are named based on their original layer and appended with a number. If you explode the block in AutoCAD 2007 or earlier releases, and then open the drawing in AutoCAD 2008 or later releases, each scale representation becomes a separate annotative object, each with one annotation scale. It is not recommended that you edit or create objects on these layers when working with a drawing created in AutoCAD 2008 and later releases in AutoCAD 2007 and earlier releases.

When this option is not selected, a single model space representation is displayed on the Model tab. More annotation objects may be displayed on the Model tab depending on the ANNOALLVISIBLE setting. Also, more objects may be displayed in paper space viewports at different sizes than in AutoCAD 2008 and later releases.

For a procedure to set this option for annotative objects, see To maintain visual fidelity for annotative objects on page 232.

See also:

- Work with Annotative Styles on page 1401

Quick Reference

Commands

ATTDEF

Creates an attribute definition for storing data in a block.

BLOCK

Creates a block definition from selected objects.

DIMSTYLE

Creates and modifies dimension styles.

HATCH

Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

MLEADERSTYLE

Creates and modifies multileader styles.

MTEXT

Creates a multiline text object.

STYLE

Creates, modifies, or specifies text styles.

TEXT

Creates a single-line text object.

System Variables

No entries

Utilities

No entries

Command Modifiers

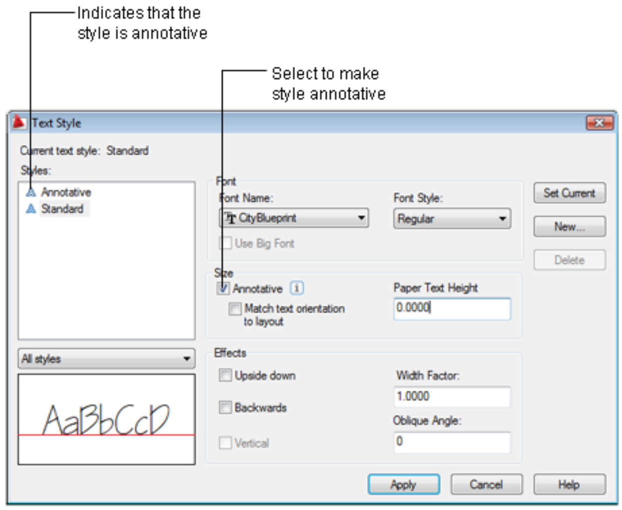
No entries

Work with Annotative Styles

You can minimize the steps to annotate a drawing by using annotative styles.

Annotative text, dimension, and multileader styles create objects.

The dialog boxes used to define these objects contain an Annotative check box where you can make the styles annotative. Annotative styles display a special  icon before their names in dialog boxes and the Properties palette.



You should specify the Paper Height value for any annotative text styles you create. The Paper Height setting specifies the height of the text in paper space.

NOTE If you've specified the Paper Height value for a dimension or multileader style, this setting overrides the text style Paper Height setting.

If you redefine styles to be annotative or nonannotative, existing objects that reference those styles are not automatically updated to reflect the annotative property of the style or definition. Use the ANNOUPDATE command to update the existing objects to the current Annotative properties of the style.

When you change the Style property of an existing object (whether it's annotative or nonannotative), the object's annotative properties will match that of the new style. If the style does not have a fixed height (the Height value is 0), the paper height of the object is calculated based on the object's current height and the annotation scale.

See also:

- Work with Text Styles on page 1544
- Create Annotative Text on page 1404
- Use Dimension Styles on page 1630
- Create Annotative Dimensions and Tolerances on page 1409

- Work with Leader Styles on page 1525
- Create Annotative Leaders and Multileaders on page 1413

To create an annotative style

- Follow the steps in one of the following procedures
  - To create a new annotative text style on page 1405
  - To change an existing nonannotative text style to annotative on page 1405
  - To create a new annotative dimension style on page 1410
  - To change an existing dimension style to annotative on page 1410
  - To create a new annotative multileader style on page 1414
  - To change an existing multileader style to annotative on page 1414

To create annotative objects from annotative styles

- Follow the steps in one of the following procedures
  - To create annotative single-line text on page 1406
  - To create annotative multiline text on page 1406
  - To create an annotative dimension on page 1411
  - To create an annotative multileader on page 1415

Quick Reference

Commands

ANNOUPDATE

Updates existing annotative objects to match the current properties of their styles.

DIMSTYLE

Creates and modifies dimension styles.

MLEADERSTYLE

Creates and modifies multileader styles.

STYLE

Creates, modifies, or specifies text styles.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Create Annotative Text

Use text for notes and labels in your drawing. You create annotative text by using an annotative text style, which sets the height of the text on the paper.

The current automatically determines the display size of the text in model space or paper space viewports.

For example, you want text to display at a height of 3/16" on the paper, so you can define a text style to have a paper height of 3/16". When you add text to a viewport that has a scale of 1/2"=1'0", the current annotation scale, which is set to the same scale as the viewport's, automatically scales the text to display appropriately at 4.5".

You can also change existing nonannotative text to annotative by changing the text's Annotative property to Yes. This applies to any text created through text styles or through the TEXT and MTEXT commands.

You can set the orientation of annotative text objects to match the orientation of the paper. For more information about setting the orientation of annotative objects, see Set Orientation for Annotations on page 1428.

See also:

- Create Text on page 1472
- Work with Annotative Styles on page 1401

To create a new annotative text style

1 Click Annotate tab ➤ Text panel ➤ Text Style.

2 In the Text Style dialog box, click New.

3 In the New Text Style dialog box, enter a new style name.

4 Click OK.

5 In the Text Style dialog box, under Size, select Annotative.

6 In the Paper Text Height box, enter the height of the text as it will display on paper.

7 Click Apply.

8 (Optional) Click Set Current to set this style as the current text style.

9 Click Close.

Command entry: STYLE

To change an existing nonannotative text style to annotative

1 Click Annotate tab ➤ Text panel ➤ Text Style.

2 In the Text Style dialog box, Styles list, select a style.

NOTE A icon next to a text style name indicates that the style is already annotative.

---

3 Under Size, select Annotative.

4 In the Paper Text Height box, enter the height of the text as it will display on paper.

5 Click Apply.

6 (Optional) Click Set Current to set this style as the current text style.

7 Click Close.

Command entry: STYLE

To create annotative single-line text

1. Click Annotate tab ➤ Text panel ➤ Text Style.
2. In the Text Style dialog box, Styles list, select an annotative text style.

   NOTE A icon next to a text style name indicates that the style is annotative.

   ---

3. Click Set Current to set this style as the current text style.
4. Click Close.
5. Click Annotate tab ➤ Text panel ➤ Single Line Text.
6. Specify the insertion point for the first character.
7. Specify a text rotation angle.
8. Enter the text.

Command entry: STYLE, TEXT

To create annotative multiline text

1. Click Annotate tab ➤ Text panel ➤ Multiline Text.
2. Specify opposite corners of a bounding box to define the width of the multiline text object. The In-Place Text Editor is displayed.
3. Do one of the following:
   - On the Text Formatting toolbar, in the Text Style control, click the arrow and select an existing annotative text style from the list.
   - Click the Annotative button on the toolbar to create annotative multiline text.
4. Enter the text.

5 On the Text Formatting toolbar, click OK.

Command entry: MTEXT

To change existing multiline text to annotative or nonannotative

1 Double-click a multiline text object. The In-Place Text Editor is displayed.

2 Click the Annotative button on the toolbar to change existing multiline text to annotative or nonannotative.

  When the Annotative button is depressed, the text is annotative. When the button is not depressed, the text is nonannotative.

3 Click OK to save the changes.

Shortcut menu: Select and right-click the text. Click Properties to change the annotative property of the text.

To change existing text (single-line or multiline) to be annotative or nonannotative

1 In the drawing, select a text object.

2 Click View tab ➤ Palettes panel ➤ Properties.

3 In the Properties palette, under Text, click Annotative.

4 On the drop-down list, select Yes or No.

Command entry: PROPERTIES

To update text to reflect the current annotative properties of the text style

1 Click Annotate tab ➤ Text panel ➤ Text Style.

2 In the Text Style dialog box, Styles list, select the style used by the text that you want to update.

NOTE A icon next to a text style name indicates that the style is already annotative.

---

3 Under Size, select Annotative.
4 In the Paper Text Height box, enter the height of the text as it will display on paper.
5 Click Apply.
6 (Optional) Click Set Current to set this style as the current text style.
7 Click Close.
8 In the drawing, select all the text objects (text and mtext) that you want to update.
9 At the command prompt, enter **annoupdate**.

Command entry: STYLE

To change the height of annotative text as it will display on the paper

1 In the drawing, select a text object.
2 Click View tab ➤ Palettes panel ➤ Properties.
3 In the Properties palette, under Paper Text Height, enter a new value.

Command entry: PROPERTIES

Quick Reference

Commands

MTEXT

Creates a multiline text object.

STYLE

Creates, modifies, or specifies text styles.

TEXT

Creates a single-line text object.

System Variables

No entries

Utilities

No entries

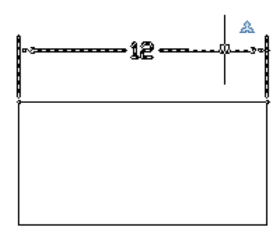
Command Modifiers

No entries

Create Annotative Dimensions and Tolerances

You can create dimensions for measurements in your drawing through annotative dimension styles.

Annotative dimension styles create dimensions in which all the elements of the dimension, such as text, spacing, and arrows, scale uniformly by the .



If you associate a dimension to an annotative object, the associativity of the dimension is lost.

You can also change an existing nonannotative dimension to annotative by changing the dimension's Annotative property to Yes.

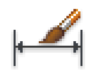
NOTE When the current dimension style is annotative, the value of DIMSCALE is automatically set to zero, and does not affect the dimension scale.

You can also create annotative tolerances. Geometric tolerances show acceptable deviations of form, profile, orientation, location, and runout of a feature.

See also:

- Dimensions and Tolerances on page 1623
- Use Dimension Styles on page 1630
- Work with Annotative Styles on page 1401

To create a new annotative dimension style

1 Click Annotate tab ➤ Dimensions panel ➤ Dimension Style. 

2 In the Dimension Style Manager dialog box, click New.

3 In the Create New Dimension Style dialog box, enter a new style name.

4 Select Annotative.

5 Click Continue.

6 In the New Dimension Style dialog box, select the appropriate tab and make changes to define the dimension style.

7 Click OK.

8 (Optional) Click Set Current to set this style as the current dimension style.

9 Click Close.

Command entry: DIMSTYLE

To change an existing dimension style to annotative

1 Click Annotate tab ➤ Dimensions panel ➤ Dimension Style. 

2 In the Dimension Style Manager dialog box, Styles list, select a style.

NOTE A icon next to a dimension style name indicates that the style is already annotative.

3 Click Modify.

4 In the Modify Dimension Style dialog box, Fit tab, under Scale for Dimension Features, select Annotative.

5 Click OK.

6 (Optional) Click Set Current to set this style as the current dimension style.

7 Click Close.

Command entry: DIMSTYLE

To create an annotative dimension

1 Click Annotate tab ➤ Dimensions panel ➤ Dimension Style.

2 In the Dimension Style Manager dialog box, Styles list, select an annotative dimension style.

A icon next to a dimension style name indicates that the style is annotative.

3 Click Set Current.

4 Click Close.

5 Click Dimension menu and select a dimension type.

6 Press ENTER to select the object to dimension or specify the first and second extension line origins.

7 Specify the dimension line location.

Command entry: DIMSTYLE

To change an existing dimension to annotative or nonannotative

1 Select a dimension in a drawing.

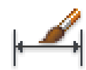
2 Click View tab ➤ Palettes panel ➤ Properties. 

3 In the Properties palette, under Misc, click Annotative.

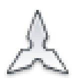
4 On the drop-down list, select Yes or No.

Command entry: PROPERTIES

To update dimensions to reflect the current annotative properties of the dimension style

1 Click Annotate tab ➤ Dimensions panel ➤ Dimension Style. 

2 In the Dimension Style Manager dialog box, Styles list, select a style.

NOTE A  icon next to a dimension style name indicates that the style is already annotative.

---

3 Click Modify.

4 In the Modify Dimension Style dialog box, Fit tab, under Scale for Dimension Features, select Annotative.

5 Click OK.

6 (Optional) Click Set Current to set this style as the current dimension style.

7 Click Close.

8 In the drawing, select all the dimensions that you want to update.

9 At the command prompt, enter **annoupdate**.

Command entry: DIMSTYLE

To create an annotative tolerance

1 Follow the steps in To create geometric tolerances on page 1738.

2 Click View tab ➤ Palettes panel ➤ Properties.

3 In the Properties palette, under Misc, click Annotative.

4 On the drop-down list, select Yes or No.

Command entry: PROPERTIES

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

STYLE

Creates, modifies, or specifies text styles.

System Variables

DIMANNO

Creates a single-line text object.

Utilities

No entries

Command Modifiers

No entries

Create Annotative Leaders and Multileaders

Leaders and multileader on page 1941 are used to add call outs to your drawings. You can create leaders through an annotative dimension style and multileaders through an annotative multileader style.

When you create a leader, you create two separate objects: the leader and the text, block, or tolerance associated with the leader. When you create a multileader, you create a single object.

If the multileader style is annotative, the associated text or tolerance will be annotative as well, regardless of the annotative setting of the text style or tolerance.

NOTE It is recommended that you create non-annotative entities when creating a mleader content block.

Blocks used in leaders and multileaders must be non-annotative.

You can change the Annotative property of leaders and multileaders in the Properties palette.

See also:

- Create Leaders on page 1514
- Work with Leader Styles on page 1525

To create a new annotative multileader style

1. Click Annotate tab ➤ Multileaders panel ➤ Multileader Style. 
2. In the Multileader Style Manager dialog box, click New.
3. In the Create New Multileader Style dialog box, enter a new style name, and select Annotative.
4. Click Continue.
5. In the Modify Multileader Style dialog box, select the appropriate tab and make changes to define the multileader style.
6. Click OK.
7. (Optional) Click Set Current to set this style as the multileader style.
8. Click Close.

Command entry: MLEADERSTYLE

To change an existing multileader style to annotative

1. Click Annotate tab ➤ Multileaders panel ➤ Multileader Style. 

2 In the Multileader Style Manager dialog box, Styles list, select a style.

NOTE A icon next to a multileader style name indicates that the style is already annotative.

---

3 Click Modify.

4 In the Modify Multileader Style dialog box, Leader Structure tab, under Scale, select Annotative.

5 Click OK.

6 (Optional) Click Set Current to set this style as the multileader style.

7 Click Close.

Command entry: MLEADERSTYLE

To create an annotative multileader

1 Click Annotate tab ➤ Multileaders panel ➤ Multileader Style.

2 In the Multileader Style Manager dialog box, Styles list, select an annotative multileader style.

NOTE A icon next to a multileader style name indicates that the style is annotative.

---

3 Click Set Current.

4 Click Close.

5 Click Annotate tab ➤ Multileaders panel ➤ Multileader.

6 Choose a point for the leader head.

7 Choose the last point for the leader.

8 Specify the text width.

9 Enter text.

10 On the Text Formatting toolbar, click OK.

Command entry: MLEADERSTYLE, MLEADER

To change an existing leader or multileader to annotative or nonannotative

1 Select a leader or multileader in a drawing.

2 Click View tab ➤ Palettes panel ➤ Properties.

3 In the Properties palette, under Misc, click Annotative.

4 On the drop-down list, select Yes or No.

Command entry: PROPERTIES

Quick Reference

Commands

MLEADERSTYLE

Creates and modifies multileader styles.

STYLE

Creates, modifies, or specifies text styles.

System Variables

No entries

Utilities

No entries

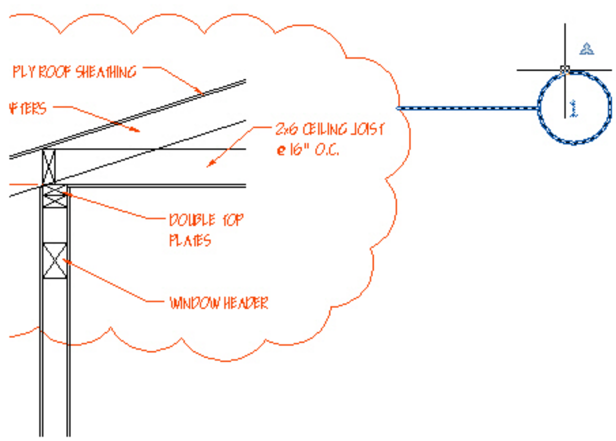
Command Modifiers

No entries

Create Annotative Blocks and Attributes

If you want to use geometric objects to annotate your drawing, combine the objects into an annotative block definition.

block definitions create annotative block references. Annotative block references and attributes initially support the current annotation scale at the time they are inserted. You should insert annotative block references with a unit factor of 1.



You cannot change the Annotative property of individual block references.

To set an annotative block's paper size, you should define the block in paper space or on the Model tab with the set to 1:1.

When creating and working with annotative blocks and annotative objects within blocks, the following points should be noted:

- Nonannotative blocks can contain annotative objects, which are scaled by the block's scale factor in addition to the annotation scale.
- Annotative blocks cannot reside in annotative blocks.
- Annotative block references are scaled uniformly by the current annotation scale as well as any user scale applied to the block reference.
- Blocks that contain annotative objects should not be manually scaled.

You can define annotative attributes for annotative and nonannotative blocks. Use annotative attributes with nonannotative blocks when you want the geometry in the block to display on the paper based on the scale of the viewport, but you want the attribute text to display at the Paper Height defined for the attribute.

You can set the orientation of annotative blocks to match the orientation of the paper. For more information about setting the orientation of annotative objects, see Set Orientation for Annotations on page 1428.

You can use the ANNOTATIVEDWG system variable to specify whether or not the entire drawing will behave as an annotative block when inserted into another drawing. The ANNOTATIVEDWG system variable becomes read-only if the drawing contains annotative objects.

NOTE The INSUNITS setting is ignored when inserting blocks into a drawing.

See also:

- Create and Use Blocks (Symbols) on page 853
- Attach Data to Blocks (Block Attributes) on page 1027

To create an annotative block definition

1. Click Blocks & References tab ➤ Block panel ➤ Create. 
2. In the Block Definition dialog box, enter a block name in the Name box.
3. Under Objects, select Convert to Block.
4. Click Select Objects.

5 Under Behavior, select Annotative.

6 Use your pointing device to select objects to be included in block definition. Press ENTER to complete object selection.

7 In the Block Definition dialog box, under Base Point, specify the block insertion point.

8 Click OK.

Command entry: BLOCK

To update existing block references to be annotative

1 Click Blocks & References tab ➤ Block panel ➤ Create. 

2 In the Block Definition dialog box, Name box, click the arrow and select the name of the block you want to update to be annotative.

3 Under Behavior, select Annotative.

4 Click OK.

The existing block references in the drawing are now annotative.

Command entry: BLOCK

To create an annotative attribute definition

1 Click Blocks & References tab ➤ Attributes panel ➤ Define Attributes.

2 In the Attribute Definition dialog box, set the Attribute Modes and enter Tag information, Insertion Point and Text Settings.

3 Under Text Settings, select Annotative.

4 Click OK.

5 Specify the start point.

6 Press ENTER.

Command entry: ATTDEF

Quick Reference

Commands

ATTDEF

Creates an attribute definition for storing data in a block.

BLOCK

Creates a block definition from selected objects.

System Variables

ANNOTATIVEDWG

Specifies whether or not the drawing will behave as an annotative block when inserted into another drawing.

Utilities

No entries

Command Modifiers

No entries

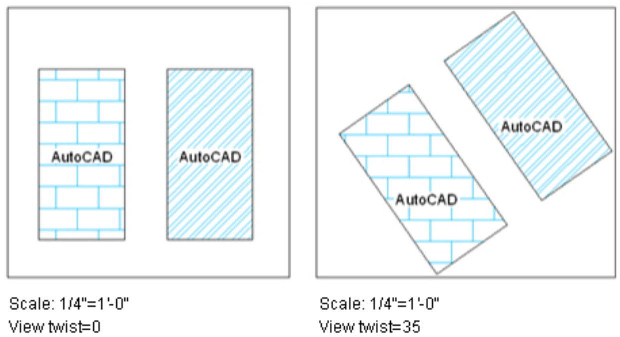
Create Annotative Hatches

Use an annotative hatch to symbolically represent material such as sand, concrete, steel, earth, etc.

An hatch is defined at a paper size. You can create individual annotative hatch objects as well as annotative hatch patterns.

The hatch pattern definitions stored in the *acad.pat* file contain information that indicates whether the pattern is annotative or non-annotative. When the selected hatch pattern is annotative, the Annotative checkbox in the Hatch and Gradient dialog box should be selected.

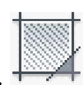
The orientation of annotative hatches always matches the orientation of the layout.



See also:

- Overview of Hatch Pattern Definitions in the *Customization Guide*

To create an annotative hatch object

1 Click Home tab ➤ Draw panel ➤ Hatch. 

2 In the Hatch and Gradient dialog box, click Add: Select Objects.

3 Specify the object or objects you want to hatch.

4 Under Options, select Annotative.

5 Click OK.

Command entry: HATCH

To change an existing hatch object to annotative

1 In model space, at the command prompt, enter **cannoscale**.

2 Enter the scale set for the viewport in which the hatch is displayed.

3 In the drawing, select the hatch.

4 In the Properties palette, under Pattern, click Annotative.

5 On the drop-down list, select Yes.

Quick Reference

Commands

HATCH

Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Display Annotative Objects

For model space or a layout viewport, you can display all the annotative objects or only those that support the current annotation scale.

This reduces the need to use multiple layers to manage the visibility of your annotations.

You use the Annotation Visibility button on the right side of the application or drawing status bar to choose the display setting for annotative objects.

Annotation visibility is turned on by default.  When annotation visibility is turned on, all annotative objects are displayed. When annotation visibility is turned off , only annotative objects for the current scale are displayed.

In general, you should turn off annotation visibility, except when inspecting a drawing created by another person or when adding scales to existing annotative objects.

Annotation visibility is also controlled by the ANNOALLVISIBLE system variable.

In order for an annotative object to be visible, the layer the object is on must be turned on.

If an object supports more than one annotation scale, the object will display at the current scale.

When the MSLTSCALE system variable is set to 1 (default), linetypes displayed on the model tab are scaled by the annotation scale

See also:

■ Drawing Status Bar on page 37

To display or hide annotative objects in a drawing

➤ On the drawing or application status bar, click the Annotation Visibility button.

When the  button is displayed, all annotative objects are displayed.

When the  button is displayed, only annotative objects that support the current annotation scale are displayed.

Quick Reference

Commands

No entries

System Variables

ANNOALLVISIBLE

Hides or displays annotative objects that do not support the current annotation scale.

MSLTSCALE

Scales linetypes displayed on the model tab by the annotation scale.

SELECTIONANNODISPLAY

Controls whether alternate scale representations are temporarily displayed in a dimmed state when an annotative object is selected.

Utilities

No entries

Command Modifiers

No entries

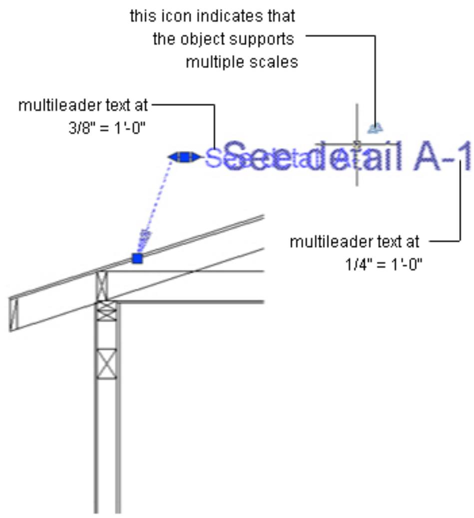
Add and Modify Scale Representations

When you create an object in your drawing, it supports one : the annotation scale that was current when you created the object. You can update annotative objects to support additional annotation scales.

When you update an annotative object to support additional scales, you add additional to the object.

For example, if an annotative multileader supports two annotations scales, it has two scale representations.

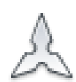
When you select an annotative object, grips are displayed on the scale representation that supports the current annotation scale. You can use these grips to manipulate the current scale representation. All other scale representations of the object are displayed in a dimmed state when the SELECTIONANNODISPLAY system variable is set to 1 (default).



Use the ANNORESET command to reset the location of all scale representations for an annotative object to that of the current scale representation.

To add the current annotation scale to an annotative object

1 Click Annotate tab ➤ Annotation Scaling panel ➤ Add/Delete Scales.



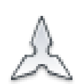
2 In a drawing, select one or more annotative objects.

3 Press ENTER.

Command entry: OBJECTSCALE
Shortcut menu: With an annotative object selected, right-click in the drawing area. Click Annotative Object Scale ➤ Add Current Scale.

To delete the current annotation scale from an annotative object

1 Click Annotate tab ➤ Annotation Scaling panel ➤ Add/Delete Scales.

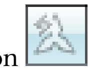


2 In a drawing, select one or more annotative objects.

3 Press ENTER.

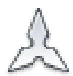
Command entry: OBJECTSCALE
Shortcut menu: With an annotative object selected, right-click in the drawing area. Click Annotative Object Scale ➤ Add Current Scale.

To automatically update annotative objects to support the current annotation scale

- On the drawing or application status bar, click the button  so it displays as  .

To add an annotation scale to an annotative object

1. Click Annotate tab ➤ Annotation Scaling panel ➤ Add/Delete Scales.

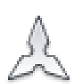
   

2. In the drawing area, select one or more annotative objects.
3. Press ENTER.
4. In the Annotative Object Scale dialog box, click Add.
5. In the Add Scales to Object dialog box, select one or more scales to add to the objects. (Press and hold the SHIFT key to select more than one scale.)
6. Click OK.
7. In the Annotative Object Scale dialog box, click OK.

Command entry: OBJECTSCALE
Shortcut menu: With an annotative object selected, right-click in the drawing area. Click Annotative Object Scale ➤ Add/Delete Scales.

To delete an annotation scale from an annotative object

1. Click Annotate tab ➤ Annotation Scaling panel ➤ Add/Delete Scales.

2. In the drawing area, select one or more annotative objects.

3 Press ENTER.

4 In the Annotative Object Scale dialog box, select one or more scales to delete from the objects. (Press and hold the SHIFT key to select more than one scale.).

NOTE You cannot delete the 1:1 scale.

5 Click OK.

Command entry: OBJECTSCALE
Shortcut menu: With an annotative object selected, right-click in the drawing area. Click Annotative Object Scale ➤ Add/Delete Scales.

Quick Reference

Commands

ANNORESET

Resets the locations of all alternate scale representations of the selected annotative objects.

OBJECTSCALE

Adds or deletes supported scales for annotative objects.

System Variables

SELECTIONANNODISPLAY

Controls whether alternate scale representations are temporarily displayed in a dimmed state when an annotative object is selected.

Utilities

No entries

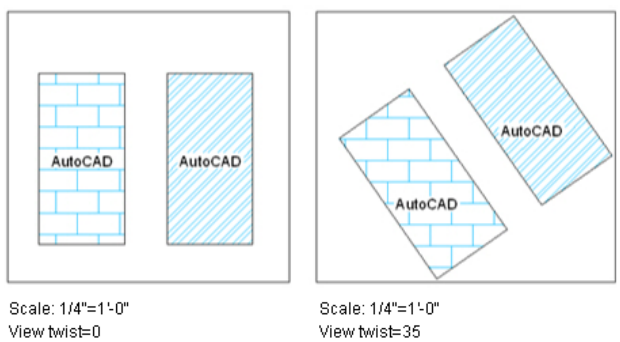
Command Modifiers

No entries

Set Orientation for Annotations

blocks and text can be set so that their orientation matches the orientation of the layout. The orientation of annotative hatches always matches the orientation of the layout.

Even if the view in the layout viewport is twisted or if the viewpoint is non-planar, the orientation of these objects in layout viewports will match the orientation of the layout.



Annotative attributes in blocks match the paper orientation of the block.

See also:

- Work with Text Styles on page 1544
- Create Annotative Text on page 1404
- Create Annotative Blocks and Attributes on page 1417

- Create Annotative Hatches on page 1420

To match the layout's orientation for an annotative text style

1. Click Annotate tab ➤ Text panel ➤ Text Style.
2. In the Text Style dialog box, Styles list, select an annotative text style.

NOTE A icon next to a text style name indicates that the style is annotative.

---

3. Under Size, select Match Text Orientation to Layout.
4. Click Apply.
5. Click Close.

Command entry: STYLE

To match the layout's orientation for an annotative block definition

1. Click Blocks & References tab ➤ Block panel ➤ Create.
2. In the Block Definition dialog box, under Name, select a block.
3. Under Behavior, select Annotative.
4. Under Behavior, select Match Block Orientation to Layout.
5. Click Close.

Command entry: BLOCK

To match the layout's orientation for an existing annotative text object

1. In the drawing, select an annotative text object.
2. Click Annotate tab ➤ Text panel ➤ Text Style.

3 In the Text Style dialog box, under Size, select Match Orientation to Layout.

Command entry: STYLE

Hatches, Fills, and Wipeouts

24

Overview of Hatch Patterns and Fills

Define the Boundaries of a Hatch

You can choose from several methods to specify the boundaries of a hatch.

- Specify a point in an area that is enclosed by objects.
- Select objects that enclose an area.
- Drag a hatch pattern into an enclosed area from a tool palette or DesignCenter.

When you hatch a drawing, whole or partial objects that are not part of the object boundary are ignored.

If a hatch line encounters an object such as text, an attribute, or a solid-fill object, and if the object is selected as part of the boundary set, HATCH hatches around the object.

text object not part of the boundary set

text object included in the boundary set

NOTE If you want to hatch an area whose boundary is not quite closed, you can set the HPGAPTOL system variable to bridge gaps and treat the boundary as closed. HPGAPTOL applies only to gaps between lines and arcs that, if extended, would meet.

To reduce file size, a hatched area is defined in the drawing database as a single graphical object.

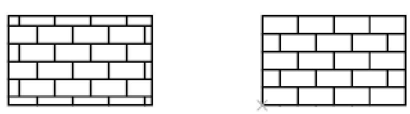
Add Hatch Patterns and Solid Fills

You can use several methods to add hatch patterns to your drawing.

- The HATCH command provides the most options.
- You can drag hatches from a tool palette. Use tool palettes when you need additional speed and convenience.
  With the Tool Palettes window open, you can right-click a pattern tool to access the Tool Properties dialog box from the shortcut menu. This dialog box contains several hatch pattern options that are also available through HATCH. For example, you can specify the scale and spacing for the hatch pattern.
- You can also use DesignCenter.

Control the Hatch Origin

By default, hatch patterns always "line up" with each other. However, sometimes you might need to move the starting point, called the *origin point*, of the hatch. For example, if you create a brick pattern, you might want to start with a complete brick in the lower-left corner of the hatched area. In that case, use the Hatch Origin options in the Hatch and Gradient dialog box.



default hatch origin

new hatch origin

The location and behavior of a hatch pattern depends on the HPORIGIN, HPORIGINMODE, and HPINHERIT system variables, and the location and orientation of the user coordinate system.

Choose a Hatch Pattern

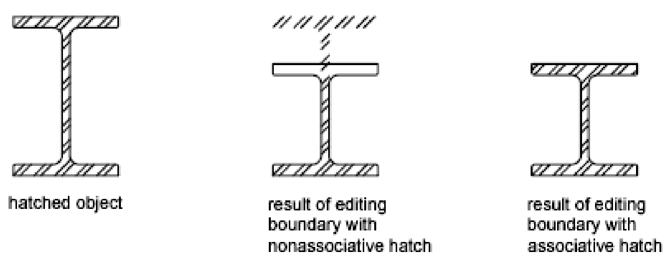
The program supplies a solid fill and more than 50 industry-standard hatch patterns that you can use to differentiate the components of objects or represent object materials. The program also comes with 14 hatch patterns that conform to the ISO (International Standards Organization) standards. When you select an ISO pattern, you can specify a pen width, which determines the lineweight in the pattern.

On the Hatch of the Hatch and Gradient dialog box, the Type and Pattern area displays the names of all the hatch patterns defined in the *acad.pat* text file. You can add new hatch patterns to the dialog box by adding their definitions to the *acad.pat* file.

Create Associative Hatches

An *associative* hatch is updated when you change the boundary. Hatched areas created with HATCH are associative by default. This setting is stored in the system variable HPASSOC. Hatches created by dragging hatch patterns from tool palettes or DesignCenter™ use the setting in HPASSOC. You can remove hatch associativity at any time or use HATCH to create a nonassociative hatch. When the HPGAPTOL system variable is set to 0 (the default), associativity is automatically removed if editing creates an open boundary.

You can use HATCH to create nonassociative hatches, which are independent of their boundaries.



Create an Annotative Hatch

An hatch is defined at a paper size. You can create individual annotative hatch objects as well as annotative hatch patterns.

Use an annotative hatch to symbolically represent material such as sand, concrete, steel, earth, etc.

For more information about creating and working with an annotative hatch, see Create Annotative Hatches on page 1420.

Assign a Draw Order to a Hatch

You can assign a draw order to a hatch so that it is drawn either behind or in front of the hatch boundary, or behind or in front of all other objects.

When you create a hatch, by default the hatch is drawn behind the hatch boundary. This makes it easier to view and select the hatch boundary. You can change the draw order of the hatch so that the hatch is drawn in front of the hatch boundary instead, or either behind or in front of all other objects. This setting is stored in the HPDRAWORDER system variable. Hatches created by dragging hatch patterns from tool palettes or DesignCenter use the draw order setting in HPDRAWORDER.

Limit Hatch Pattern Density

If you create a very dense hatch, the program may reject the hatch and display a message indicating that the hatch scale is too small or its dash length too short. You can change the maximum number of hatch lines by setting the HPMAXLINES system variable. You can set values between 100 and 10000000 (ten million). The default value for HPMAXLINES is 1000000.

Edit Hatch Boundaries

Because there are so many combinations of objects that can be hatched, editing hatched geometry can produce unexpected results. If you create a hatch that you don't want, you can undo it, trim the hatch, or delete the hatch and rehatch the area.

Create Custom Hatch Patterns

You can also define your own hatch pattern using the current linetype with the User Defined Pattern option of the Hatch and Gradient dialog box, or you can create more complex hatch patterns.

See also:

- Modify Hatches and Solid-Filled Areas on page 1459
- “Overview of Hatch Pattern Definitions” in the *Customization Guide*
- Scale Annotations on page 1393

To drag hatch patterns into your drawing

1 Click Insert tab ➤ Content panel ➤ Design Center.

NOTE This procedure describes how to use DesignCenter to drag hatch patterns into your drawing. You can also drag hatch patterns from a tool palette.

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2 On the Folders , click the Search button.

3 In the Search dialog box, make the following entries:

- In Look For, select Hatch Pattern Files.
- In the In box, select the drive where the program is installed.
- Select the Search Subfolders option.
- On the Hatch Pattern Files , in Search for the Name, enter ***** (asterisk).

4 Click Search Now.

The default hatch pattern file is *acad.pat* or *acadiso.pat*. The search results may display the same file in different locations.

NOTE For convenient access, you can add the PAT file to Favorites by selecting the file and clicking the Favorites button. A shortcut to the PAT file is displayed in the *Favorites* folder on the Folders in DesignCenter.

---

5 In the search results, double-click the file to load the hatch patterns into the content area of DesignCenter.

6 (Optional) Right-click a pattern to display a shortcut menu with the following options:

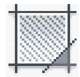
- **HATCH**. Opens the Hatch and Gradient dialog box.
- **Copy**. Stores the hatch pattern to the Clipboard.
- **Create Tool Palette**. Creates a new tool palette with the selected pattern displayed.

7 From the content area, drag a hatch pattern onto a closed object in your drawing or onto a tool palette.

NOTE If the hatch pattern scale is too large or small, an error message is displayed. You can adjust the scale for any hatch pattern by double-clicking it to display the Hatch and Gradient dialog box.

Toolbar: Standard 
Command entry: ADCENTER

To hatch areas

1 Click Home tab ➤ Draw panel ➤ Hatch. 

2 In the Hatch and Gradient dialog box, click Add: Pick points.

3 In your drawing, specify a point inside each area that you want hatched, and then press ENTER.

   This point is known as the internal point.

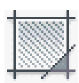
4 In the Hatch and Gradient dialog box, Hatch , in the swatch box, verify that the sample pattern is the pattern you want to use. To change patterns, select another pattern from the Pattern list.

5 In the Hatch and Gradient dialog box, make adjustments, if necessary.

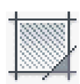
6 Under Draw Order, click one of the options.

   You can change the draw order of the hatch so that the hatch is drawn either behind or in front of the hatch boundary, or behind or in front of all other objects.

7 Click OK.

Toolbar: Draw 
Command entry: HATCH

To hatch selected objects

1 Click Home tab ➤ Draw panel ➤ Hatch. 

2 In the Hatch and Gradient dialog box, click Add: Select objects.

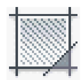
3 Specify the object or objects you want to hatch.

   The objects need not form a closed boundary. You can also specify any islands that should remain unhatched. Also, you can set the HPGAPTOL system variable to treat a set of objects that almost enclose an area as a closed hatch boundary.

4 Under Draw Order, click one of the options.

You can change the draw order of the hatch so that the hatch is drawn either behind or in front of the hatch boundary, or behind or in front of all other objects.

5 Click OK.

Toolbar: Draw 
Command entry: HATCH

Quick Reference

Commands

ADCENTER

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

BOUNDARY

Creates a region or a polyline from an enclosed area.

DONUT

Creates a filled circle or a wide ring.

FILL

Controls the filling of objects such as hatches, 2D solids, and wide polylines.

HATCH

Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

HATCHEDIT

Modifies an existing hatch or fill.

MATCHPROP

Applies the properties of a selected object to other objects.

PLINE

Creates a 2D polyline.

PROPERTIES

Controls properties of existing objects.

SOLID

Creates solid-filled triangles and quadrilaterals.

UCS

Manages user coordinate systems.

System Variables

FILLMODE

Specifies whether hatches and fills, 2D solids, and wide polylines are filled in.

HPANG

Specifies the hatch pattern angle.

HPASSOC

Controls whether hatch patterns and gradient fills are associative.

HPBOUND

Controls the object type created by the BHATCH and BOUNDARY commands.

HPDRAWORDER

Controls the draw order of hatches and fills.

HPDOUBLE

Specifies hatch pattern doubling for user-defined patterns.

HPGAPTOL

Treats a set of objects that almost enclose an area as a closed hatch boundary.

HPINHERIT

Controls the hatch origin of the resulting hatch when using Inherit Properties in HATCH and HATCHEDIT.

HPMAXLINES

Sets the maximum number of hatch lines that are generated in a hatch operation.

HPNAME

Sets a default hatch pattern name of up to 34 characters without spaces.

HPOBJWARNING

Sets the number of hatch boundary objects that can be selected before displaying a warning message.

HPINHERIT

Controls the hatch origin of the resulting hatch when using Inherit Properties in HATCH and HATCHEDIT.

HPORIGIN

Sets the hatch origin point for new hatch objects relative to the current user coordinate system.

HPORIGINMODE

Controls how HATCH determines the default hatch origin point.

HPSCALE

Specifies the hatch pattern scale factor, which must be greater than zero.

HPSEPARATE

Controls whether HATCH creates a single hatch object or separate hatch objects when operating on several closed boundaries.

HPSPACE

Specifies the hatch pattern line spacing for user-defined simple patterns, which must be greater than zero.

PICKSTYLE

Controls the use of group selection and associative hatch selection.

Utilities

No entries

Command Modifiers

No entries

Define Hatch Boundaries

You create a hatch by selecting an object to hatch or fill, or by defining a boundary and then specifying an internal point.

Overview of Hatch Boundaries

You can hatch an enclosed area or hatch within a specified boundary using HATCH. By default, HATCH creates associative hatches that are updated when the boundary is changed.

You create a hatch by selecting an object to hatch or by defining a boundary and then specifying an internal point. A hatch boundary can be any combination of objects, such as lines, arcs, circles, and polylines, that forms an enclosed area.

Enclosed areas within the hatch area are referred to as islands. You can hatch them or leave them unhatched depending on the Islands setting in the Hatch and Gradient dialog box.

If you are hatching a small area in a complex drawing, you can use boundary sets to speed the process.

Objects can be hatched only if they are in a plane parallel to the *XY* plane of the current UCS.

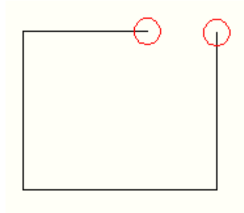
NOTE If you want to hatch an area that is not completely enclosed, you can set a gap tolerance (HPGAPTOL system variable). Any gaps equal to or smaller than the value you specify in the gap tolerance are ignored, and the boundary is treated as closed.

Invalid Hatch Boundaries

When a hatch boundary cannot be determined, it might be because the specified internal point is not within a fully enclosed area. Red circles are displayed around unconnected endpoints of the boundary to identify gaps in the hatch boundary.

NOTE If a boundary is outside a screen display, you must zoom out to ensure the point selected is within a fully enclosed area.

The red circles remain displayed even after you exit the HATCH command. They are removed when you select another internal point for the hatch, or by using the REDRAW, REGEN, or REGENALL command.



Quick Reference

Commands

BOUNDARY

Creates a region or a polyline from an enclosed area.

HATCH

Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

HATCHEDIT

Modifies an existing hatch or fill.

System Variables

HPBOUND

Controls the object type created by the BHATCH and BOUNDARY commands.

HPGAPTOL

Treats a set of objects that almost enclose an area as a closed hatch boundary.

HPOBJWARNING

Sets the number of hatch boundary objects that can be selected before displaying a warning message.

HPSEPARATE

Controls whether HATCH creates a single hatch object or separate hatch objects when operating on several closed boundaries.

PICKSTYLE

Controls the use of group selection and associative hatch selection.

Utilities

No entries

Command Modifiers

No entries

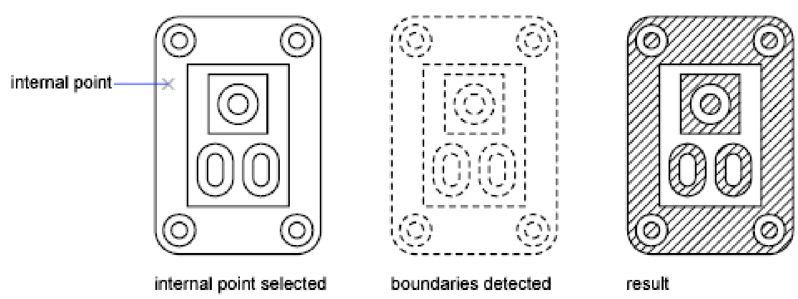
Control the Hatching in Islands

You can determine how enclosed areas within hatch boundaries, called *islands*, are hatched.

You can determine how *islands*, enclosed areas within the hatch boundary, are hatched using the three hatching styles: Normal, Outer, and Ignore. You can preview these hatching styles in the More Options area of the Hatch and Gradient dialog box.

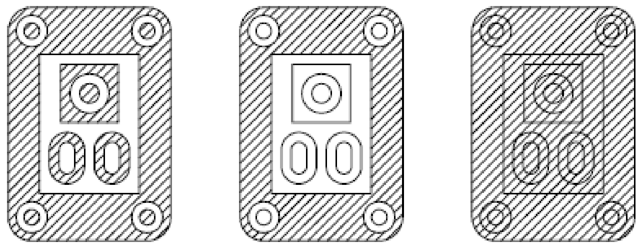
Normal hatching style (the default) hatches inward from the outer boundary. If the hatching process encounters an internal boundary, hatching is turned off until another boundary is encountered.

If you hatch using the Normal hatching style, islands remain unhatched and islands within islands are hatched, as shown below.



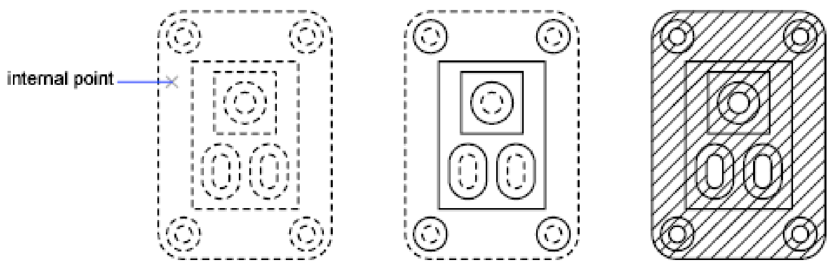
Outer hatching style hatches inward from the outer boundary and stops at the next boundary.

Ignore hatching style hatches the entire enclosed area, ignoring internal boundaries.



Normal Outer Ignore

You can also remove any islands from the hatch area.



boundaries detected

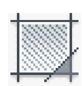
islands to remove (solid boundaries)

result

See also:

- Modify Hatches and Solid-Filled Areas on page 1459

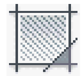
To remove islands from the hatch area

1 Click Home tab ➤ Draw panel ➤ Hatch. 

2 In the Hatch and Gradient dialog box, after adding one or more boundaries, click Remove Boundaries.

3 Select the boundaries that you want to remove and press ENTER.

4 In the Hatch and Gradient dialog box, click OK to apply the hatch.

Toolbar: Draw 
Command entry: HATCH

Quick Reference

Commands

BOUNDARY

Creates a region or a polyline from an enclosed area.

HATCH

Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

HATCHEDIT

Modifies an existing hatch or fill.

System Variables

HPOBJWARNING

Sets the number of hatch boundary objects that can be selected before displaying a warning message.

HPSEPARATE

Controls whether HATCH creates a single hatch object or separate hatch objects when operating on several closed boundaries.

Utilities

No entries

Command Modifiers

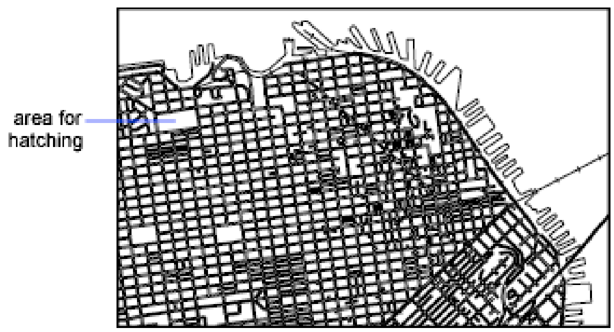
No entries

Define Hatch Boundaries in Large Drawings

You can save time hatching a small area in a complex drawing by defining a set of objects in the drawing to be used in determining the hatch boundary.

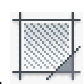
By default, HATCH defines the boundary by analyzing all closed objects in the drawing. Analysis of all objects fully or partially visible on the screen as boundaries can be time consuming in a complex drawing. To hatch a small area of a complex drawing, you can define a set of objects in the drawing called a *boundary set*. HATCH does not analyze objects that are not included in the boundary set.

For clarity, first zoom into the area you want to hatch.

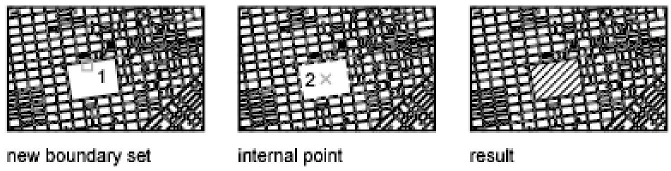


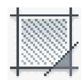
The View Selections option in the Hatch and Gradient dialog box, highlights the objects in the drawing that define the boundary.

To define a boundary set in a complex drawing

1. Click Home tab ➤ Draw panel ➤ Hatch. 
2. In the Hatch and Gradient dialog box, More Options, under Boundary Set, click New.
3. At the Select Objects prompt, specify opposite corner points for the boundary set and press ENTER.

   If you use crossing selection by specifying points from right to left, you select all objects enclosed or crossed.
4. In the Hatch and Gradient dialog box, click Add Boundary. If necessary, enter **k** to specify the Pick Internal Point option.
5. Specify the internal point.
6. Click OK to apply the hatch.



Toolbar: Draw 
Command entry: *HATCH*

Quick Reference

Commands

BOUNDARY

Creates a region or a polyline from an enclosed area.

HATCH

Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

HATCHEDIT

Modifies an existing hatch or fill.

System Variables

HPOBJWARNING

Sets the number of hatch boundary objects that can be selected before displaying a warning message.

Utilities

No entries

Command Modifiers

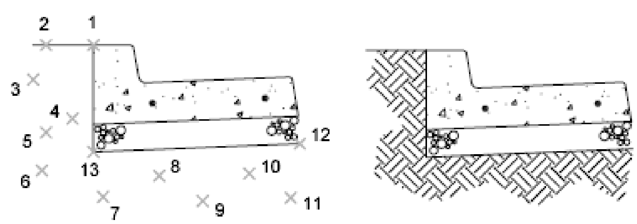
No entries

Create Unbounded Hatches

There are several methods you can use to create a hatch that does not display a hatch boundary.

- You can create a hatch with HATCH, and then erase some or all of the boundary objects.
- You can create a hatch with HATCH, making sure that the boundary objects are on a different layer than the hatch. Then turn off or freeze the layer of the boundary objects. This is the only method that maintains hatch associativity.
- You can trim an existing hatch with objects created as trim boundaries. After trimming the hatch, erase the objects.
- You can define a hatch boundary with the Draw option of -HATCH at the command prompt by specifying boundary points.

For example, you might want to show that a large area of a drawing is filled with a pattern by filling only a small section of that area, as shown in the following illustration.



points specified to define hatch boundary

result

You can choose whether to retain the polyline boundary after the hatch is created; here, the polyline boundary is not retained.

To define a boundary by specifying points

1. At the command prompt, enter **-hatch**.
2. Enter the desired pattern. For example, enter **earth** to select the EARTH pattern.
3. Specify the scale and angle for the pattern.
4. Enter **w** to specify the Draw option.
5. Specify points to define the boundary. Enter **c** to close the polyline boundary, and then press ENTER.
6. Enter **n** to discard the polyline boundary once the hatch area has been defined, or enter **y** to create a polyline.

Command entry: -HATCH

Quick Reference

Commands

HATCH

Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

HATCHEDIT

Modifies an existing hatch or fill.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Choose Hatch Patterns and Solid Fills

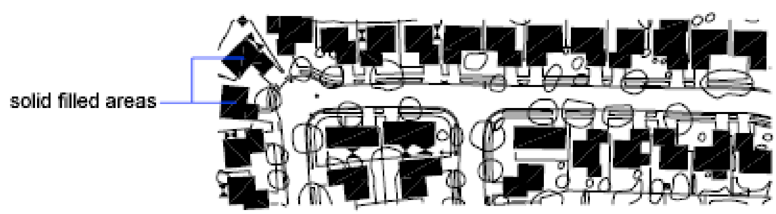
You can use a predefined hatch pattern or solid fill, or you can create your own hatch patterns.

Create Solid-Filled Areas

There are several methods for creating solid-filled areas.

Solid-filled areas can be created using

- Hatches with a solid hatch pattern (HATCH)
- 2D solids (SOLID)
- Wide polylines or donuts (PLINE, DONUT)

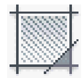


See also:

- Overview of Hatch Patterns and Fills on page 1431
- Modify Hatches and Solid-Filled Areas on page 1459
- Draw Polylines on page 801

■ Draw Donuts on page 831

To create a hatch with a solid pattern

1 Click Home tab ➤ Draw panel ➤ Hatch. 

2 In the Hatch and Gradient dialog box, click Add: Pick points.

3 Specify a point in your drawing inside each area that you want hatched. This point is known as the internal point.

4 Press ENTER.

5 In the Hatch and Gradient dialog box, Hatch , under Type, click Predefined.

6 Click the [...] button next to Pattern.

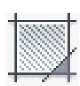
7 In the Hatch Pattern Palette dialog box, Other Predefined , select Solid. Click OK.

8 To see how the hatch pattern will look, click Preview.

9 When you finish previewing the hatch pattern, right-click or press ENTER to apply the hatch, or press any other button or key to return to the Hatch and Gradient dialog box.

10 In the Hatch and Gradient dialog box, make adjustments, if necessary. (You can specify new hatch boundaries by clicking Add Boundaries or Remove Boundaries.)

11 Click OK.

Toolbar: Draw 
Command entry: HATCH

To create a 2D solid object

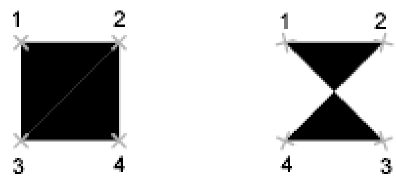
1 Click Draw menu ➤ Modeling ➤ Meshes ➤ 2D Solid.

2 Specify the first point.

3 Specify the second point, moving left to right.

4 Continue to specify points. Press ENTER when the object is complete.

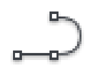
   When you create a quadrilateral solid-filled area, the sequence of the third and fourth points determines its shape. Compare the following illustrations:

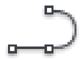
   

   Notice that to create the quadrilateral area, both the top and bottom edges are specified from left to right. If you specify the first point on the right and the second point on the left, then the third and fourth points should also be in a right-to-left direction. As you continue to specify pairs of points, be sure to continue this zigzag sequence to ensure the results you expect.

Command entry: SOLID

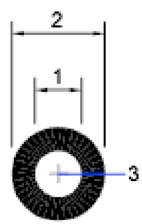
To create a wide polyline

1. Click Home tab ➤ Draw panel ➤ Polyline. 
2. Specify the start point of the line segment.
3. Enter **w** (Width).
4. Enter the starting width of the line segment.
5. Specify the ending width of the line segment using one of the following methods:
   - To create a line segment of equal width, press ENTER.
   - To create a tapering line segment, enter a different width.
6. Specify the endpoint of the polyline segment.
7. Continue specifying segment endpoints as needed.
8. Press ENTER to end, or enter **c** to close the polyline.

Toolbar: Draw 
Command entry: PLINE

To create a donut

1 Click Home tab ➤ Draw panel ➤ Donut. 
2 Specify the inside diameter (1).
3 Specify the outside diameter (2).
4 Specify the center of the donut (3).
5 Specify the center point for another donut, or press ENTER to complete the command.



Command entry: DONUT

Quick Reference

Commands

DONUT

Creates a filled circle or a wide ring.

FILL

Controls the filling of objects such as hatches, 2D solids, and wide polylines.

HATCH

Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

PLINE

Creates a 2D polyline.

SOLID

Creates solid-filled triangles and quadrilaterals.

System Variables

FILLMODE

Specifies whether hatches and fills, 2D solids, and wide polylines are filled in.

Utilities

No entries

Command Modifiers

No entries

Create Gradient-Filled Areas

A gradient fill is a solid hatch fill that gives the blended-color effect of a surface with light on it. You can use gradient fills to suggest a solid form in two-dimensional drawings.

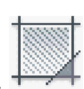
The color in a gradient fill makes a smooth transition from light to dark, or from dark to light, and back. You select a predefined pattern (for example, linear, spherical, or radial sweep) and specify an angle for the pattern. In a two-color gradient fill, the transition is both from light to dark and from the first color to the second.

Gradient fills are applied to objects in the same way solid fills are and can be associated with their boundaries or not. An associated fill is automatically updated when the boundary changes.

You cannot use plot styles to control the plotted color of gradient fills.

Double-click a gradient fill to modify it.

To create a one-color gradient fill

1 Click Home tab ➤ Draw panel ➤ Hatch. 

2 In the Hatch and Gradient dialog box, click Add: Pick points or Add: Select objects.

3 Specify an internal point or select an object, and then press ENTER.

4 In the Hatch and Gradient dialog box, Gradient , select One Color.

5 If you want to change the color, click the [...] button next to the color.

6 In the Select Color dialog box, use the Shade/Tint slider to adjust the color.

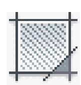
- Move the slider toward Tint to create a color transition that moves toward white.
- Move the slider toward Shade to create a transition that moves toward black.

7 Click a pattern, and set the following options:

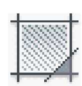
- Select Center to create a symmetrical fill, or clear Center to move the “highlight” up and to the left.
- Specify an angle for the “highlighted” area.

8 To see how the gradient fill will look, click Preview. Press ENTER or right-click to return to the dialog box and make adjustments.

9 When you are satisfied, in the Hatch and Gradient dialog box, click OK to create the gradient fill.

Toolbar: Draw 
Command entry: HATCH

To create a two-color gradient fill

1 Click Home tab ➤ Draw panel ➤ Hatch. 

2 In the Hatch and Gradient dialog box, click Add: Pick points or Add: Select objects.

3 Specify an internal point or select an object, and then press ENTER.

4 In the Hatch and Gradient dialog box, Gradient , select Two Color.

The second color is the color of the highlighted area in the gradient fill.

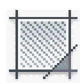
5 If you want to change either color, click the [...] button next to the color to open the Select Color dialog box.

6 Click a pattern, and set the following options:

- Select Center to create a symmetrical fill, or clear Center to move the “highlight” up and to the left.
- Specify an angle for the “highlighted” area.

7 To see how the gradient fill will look, click Preview. Press ENTER or right-click to return to the dialog box and make adjustments.

8 When you are satisfied, in the Hatch and Gradient dialog box, click OK to create the gradient fill.

Toolbar: Draw 
Command entry: HATCH

To modify a gradient fill

- Double-click a gradient fill to modify it.

Quick Reference

Commands

HATCH

Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Use Predefined Hatch Patterns

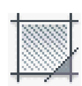
You can choose from more than 50 industry-standard hatch patterns. Additional hatch patterns are available from external pattern libraries.

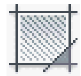
The program supplies a solid fill and more than 50 industry-standard hatch patterns, which can represent materials such as earth, brick, or clay.

Fourteen hatch patterns conform to International Standards Organization (ISO) standards. When you select an ISO pattern, you can specify a pen width, which determines the lineweight in the pattern.

In addition to using the patterns supplied with the program, you can use patterns from an external pattern library. These patterns are listed by name and are displayed in the Hatch Pattern Palette dialog box.

To use a predefined hatch pattern

1. Click Home tab ➤ Draw panel ➤ Hatch. 
2. In the Hatch and Gradient dialog box, click Add: Pick points or Add: Select objects.
3. Specify an internal point or select an object.
4. In the Hatch and Gradient dialog box, Hatch , select Predefined in the Type box.
5. From the Pattern box, select a pattern.
6. Click OK.

Toolbar: Draw 
Command entry: HATCH

Quick Reference

Commands

HATCH

Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

System Variables

HPANG

Specifies the hatch pattern angle.

HPBOUND

Controls the object type created by the BHATCH and BOUNDARY commands.

HPDOUBLE

Specifies hatch pattern doubling for user-defined patterns.

HPNAME

Sets a default hatch pattern name of up to 34 characters without spaces.

HPSCALE

Specifies the hatch pattern scale factor, which must be greater than zero.

Utilities

No entries

Command Modifiers

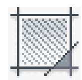
No entries

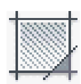
Create User-Defined Hatch Patterns

You can define a simple hatch pattern based on the current linetype.

In addition to using predefined hatch patterns, you can define a simple hatch pattern based on the current linetype. You define the pattern by changing the angle and spacing of the hatch lines.

To create a user-defined hatch pattern

1. Specify the linetype for the user-defined hatch pattern by making that linetype current.
2. Click Home tab ➤ Draw panel ➤ Hatch. 
3. In the Hatch and Gradient dialog box, click Add: Pick points or Add: Select objects.
4. Specify an internal point or select an object.
5. In the Hatch and Gradient dialog box, Hatch , select User-Defined in the Type box.
6. Specify the angle and spacing of the hatch pattern.
7. To use intersecting lines in the pattern, select Double.
8. Click OK.

Toolbar: Draw 
Command entry: HATCH

Quick Reference

Commands

HATCH

Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

System Variables

HPANG

Specifies the hatch pattern angle.

HPBOUND

Controls the object type created by the BHATCH and BOUNDARY commands.

HPDOUBLE

Specifies hatch pattern doubling for user-defined patterns.

HPNAME

Sets a default hatch pattern name of up to 34 characters without spaces.

HPSCALE

Specifies the hatch pattern scale factor, which must be greater than zero.

HPSPACE

Specifies the hatch pattern line spacing for user-defined simple patterns, which must be greater than zero.

Utilities

No entries

Command Modifiers

No entries

Modify Hatches and Solid-Filled Areas

You can modify both the pattern fill and the boundaries of hatches.

You can also modify solid-filled areas, but the method you use depends on whether the solid-filled area is a solid-filled hatch, a 2D solid, or a wide polyline or donut. You can also modify the draw order of your hatch.

Control Hatch Pattern Density

Hatching can produce a very large number of line and point objects. Although stored as hatch objects, these line and point objects do use disk space and take time to generate. If you use a relatively small scale factor when hatching an area, the hatch could require millions of line and point objects, thus taking a very long time to complete and possibly exhausting the available resources. You can avoid this problem by imposing a limit on the number of objects created by a single HATCH command. If the approximate number of objects needed for a particular hatch (considering the boundary extents, pattern, and scale) exceeds the limit, HATCH displays a message indicating that the hatch scale is too small or that its dash length is too short, and the hatch request is rejected. If this occurs, carefully examine your hatch settings. The scale factor may be unreasonable and may need to be adjusted.

The hatch object limit is set by the MaxHatch environment setting, which is stored in the system registry. Its default value is 10000. You can change this limit by setting the MaxHatch system registry variable using **(setenv "MaxHatch" "n")** where *n* is a number between 100 and 10000000 (ten million).

NOTE MaxHatch must be entered with the letters M and H capitalized and the rest of the letters in lowercase.

Change the Hatch Properties of an Existing Hatch

You can modify hatch-specific properties, such as pattern, scale, and angle, of an existing hatch. You can use the following:

- Hatch Edit dialog box (recommended)
- Properties palette

You can also copy properties from one hatch to another. With the Inherit Properties button in the Hatch Edit dialog box, you can copy all hatch-specific properties, including the hatch origin, from one hatch to another. Use the Match Properties dialog box to copy general properties and hatch-specific properties (with the exception of the hatch origin), from one hatch to another.

You can use EXPLODE to disassemble a hatch into its component objects.

Modify a Hatch Boundary

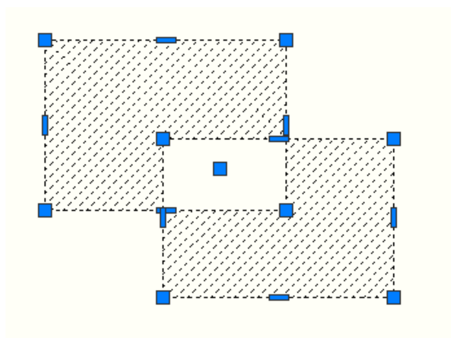
If the editing you do maintains a closed boundary, an associative hatch is updated automatically. If the editing produces an open boundary, the hatch loses any associativity with the boundary and remains unchanged. Associativity may also be lost during editing of a hatch boundary if the hatch pattern file is not available at the time of editing. If you trim a hatch and the hatch pattern (PAT) file is no longer available, the hatch will disappear.

Hatch associativity depends on whether you choose Associative in the Hatch and Gradient (HATCH) and Hatch Edit (HATCHEDIT) dialog boxes. Nonassociative hatches are not updated when their original boundary is changed.

You can remove hatch associativity at any time, but once it is removed for an existing hatch, it cannot be reestablished. The hatch must be recreated to restore associativity or a new hatch boundary must be created and associated with the hatch.

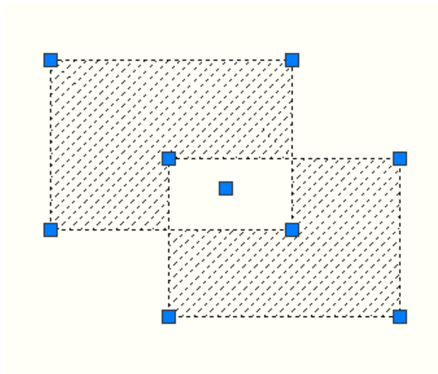
To create a boundary around a nonassociative or unbounded hatch, in the Hatch and Gradient dialog box, use the Recreate Boundary option. With this option you can also specify that the new boundary is associated with the hatch.

The Select Boundary Objects option in the Hatch Edit dialog box displays grips for a selected hatch. When you manually select or use the Select Boundary Objects option to select a nonassociative hatch, the hatch is selected and the grips display on the boundary of the hatch. You can use these grips to change the extents of nonassociative hatches.



Nonassociative hatch grips

When you manually select an associative hatch, it displays only a single grip point of the hatch. With the Select Boundary Objects option, you can select an associative hatch and display the boundary grip controls of the objects associated with it. You can extend by editing the grip controls of the associated objects.



Associative hatch grips

NOTE If a hatch is associated with XREF or BLOCK, only a single grip point is displayed. To edit the associated geometry, use REFEDIT or XOPEN.

The grip control tooltip displays more edit options available based on the grip type. You can cycle through these options by pressing the CTRL key. For example, an edge grip for a polyline segment has an option to convert the segment to an arc while an edge grip for a polyline arc segment has an option to convert to line.

Modify Solid-Filled Areas

Solid-filled areas can be represented by

- Hatches (with a solid hatch pattern)
- 2D solids
- Gradient fills
- Wide polylines or donuts

You modify each of these solid-filled objects just as you would any other hatch, 2D solid, wide polyline, or donut. In addition to PROPERTIES, you can use HATCHEDIT for solid-filled hatches and gradient fills, grip editing for 2D solids, and PEDIT for wide polylines and donuts.

Modify the Draw Order of a Hatch

When you edit a hatch, you can change its draw order, so that it is displayed behind the hatch boundary, in front of the hatch boundary, behind all other objects, or in front of all other objects.

To change the angle of a hatch

1 Select the hatch pattern.

2 Right-click the hatch. Click Properties.

3 In the Properties palette, enter the new value for Angle.

Toolbar: Standard 
Command entry: PROPERTIES

To set the limit of objects in a hatch pattern

➤ At the command prompt, enter **(setenv “MaxHatch” “n”)**

where *n* is a number between 100 and 10000000 (ten million). The higher the value, the more dense the hatch pattern.

NOTE MaxHatch is case-sensitive.

To modify a gradient fill

1 Double-click the gradient fill you want to modify.

2 In the Hatch Edit dialog box, Gradient , make any changes.

3 To see how the gradient fill will look, click Preview. Press ENTER or right-click to return to the dialog box and make adjustments.

4 When you are satisfied, in the Hatch and Gradient dialog box, click OK to create the gradient fill.

Quick Reference

Commands

EXPLODE

Breaks a compound object into its component objects.

FILL

Controls the filling of objects such as hatches, 2D solids, and wide polylines.

HATCHEDIT

Modifies an existing hatch or fill.

MATCHPROP

Applies the properties of a selected object to other objects.

PEDIT

Edits polylines and 3D polygon meshes.

PROPERTIES

Controls properties of existing objects.

System Variables

FILLMODE

Specifies whether hatches and fills, 2D solids, and wide polylines are filled in.

HPANG

Specifies the hatch pattern angle.

HPASSOC

Controls whether hatch patterns and gradient fills are associative.

HPDOUBLE

Specifies hatch pattern doubling for user-defined patterns.

HPINHERIT

Controls the hatch origin of the resulting hatch when using Inherit Properties in HATCH and HATCHEDIT.

HPNAME

Sets a default hatch pattern name of up to 34 characters without spaces.

HPORIGIN

Sets the hatch origin point for new hatch objects relative to the current user coordinate system.

HPORIGINMODE

Controls how HATCH determines the default hatch origin point.

HPSCALE

Specifies the hatch pattern scale factor, which must be greater than zero.

HPSPACE

Specifies the hatch pattern line spacing for user-defined simple patterns, which must be greater than zero.

Utilities

No entries

Command Modifiers

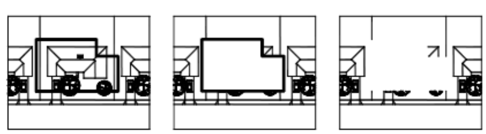
No entries

Create a Blank Area to Cover Objects

Wipeout objects cover existing objects with a blank area to make room for notes or to mask details.

A wipeout object is a polygonal area that masks underlying objects with the current background color. This area is bounded by the wipeout frame, which you can turn on for editing and turn off for plotting.

You can create a wipeout object by specifying a polygonal area with a series of points, or you can convert a closed polyline into a wipeout object.



Closed polyline created | Wipeout object created from polyline | Wipeout frame turned off

You can move or delete a wipeout object by first turning on wipeout frames so you can select it.

Requirements and Limitations

If a polyline is used to create a wipeout object, the polyline must be closed, contain line segments only, and have zero width.

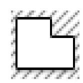
You can create wipeout objects on a layout in paper space to mask objects in model space; however, in the Page Settings dialog box, under Plot Options, the Plot Paper Space Last option must be cleared before you plot to ensure that the wipeout object is plotted correctly.

Because a wipeout object is similar to a raster image, it has the same requirements for plotting: you need a raster-capable plotter with either an ADI 4.3 raster-capable driver or the system printer driver.

See also:

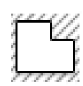
- Control How Overlapping Objects Are Displayed on page 627

To cover existing objects with a blank area

1. Click Annotate tab ➤ Markup panel ➤ Wipeout. 
2. Specify points in a sequence that defines the perimeter of the area to be masked.
3. Press ENTER to end.

Command entry: WIPEOUT

To turn all wipeout frames on or off

1. Click Annotate tab ➤ Markup panel ➤ Wipeout. 
2. At the command prompt, enter **f** (Frames).
3. Enter **on** or **off** and press ENTER.

Command entry: WIPEOUT

Quick Reference

Commands

WIPEOUT

Creates a wipeout object.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Notes and Labels

25

You can create and modify several types of text, including text with leaders. You can control most text style settings by defining text styles.

Overview of Notes and Labels

You can create text in various ways. For short, simple entries, use single-line text. For longer entries with internal formatting, use multiline text (mtext).

Although all entered text uses the current text style, which establishes the default font and format settings, you can use several methods to customize the text appearance. There are several tools that can change text scale and justification, find and replace text, and check for spelling errors.

Text that is included in a dimension or tolerance is created using the dimensioning commands. You can also create multiline text with leaders.

To create single-line text

1 Click Home tab ➤ Annotation panel ➤ Single Line Text. 

2 Specify the insertion point for the first character. If you press ENTER, the program locates the new text immediately below the last text object you created, if any.

3 Specify the height of the text. This prompt is displayed only if text height is set to 0 in the current text style.

A rubber-band line is attached from the text insertion point to the cursor. Click to set the height of the text to the length of the rubber-band line.

4 Specify a text rotation angle.

You can enter an angle value or use your pointing device.

5 Enter the text. At the end of each line, press ENTER. Enter more text as needed.

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.

If you specify another point during this command, the cursor moves to that point, and you can continue typing. Every time you press ENTER or specify a point, a new text object is created.

6 Press ENTER on a blank line to end the command.

Command entry: TEXT

To create multiline text

1 Click Home tab ➤ Annotation panel ➤ Multiline Text.

2 Specify opposite corners of a bounding box to define the width of the multiline text object.

   If the ribbon is active, the MTEXT ribbon contextual tab displays. If the ribbon is not active, the In-Place Text Editor is displayed.

3 To indent the first line of each paragraph, drag the first-line indent slider on the ruler. To indent the other lines of each paragraph, drag the paragraph slider.

4 To set tabs, click the ruler where you want a tab stop.

5 If you want to use a text style other than the default, on the ribbon, click the Annotate tab, Text panel. Select the desired text style from the drop-down list.

6 Enter text.

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.

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7 To override the current text style, select text as follows:

- To select one or more letters, click and drag the pointing device over the characters.
- To select a word, double-click the word.
- To select a paragraph, triple-click the paragraph.

8 On the ribbon, make format changes as follows:

- To change the font of the selected text, select a font from the list.
- To change the height of the selected text, enter a new value in the Height box.

  NOTE The MText height value is reset to 0 if its default height is not modified during creation.

- To format text in a TrueType font with boldface or italics, or to create underlined or overlined text for any font, click the corresponding button on the ribbon. SHX fonts do *not* support boldface or italics.
- To apply color to selected text, choose a color from the Color list. Click Other to display the Select Color dialog box.

9 To save your changes and exit the editor, use one of the following methods:

- On the MTEXT ribbon contextual tab, in the Close panel, click Close Text Editor.
- Click in the drawing outside the editor.
- Press CTRL+ENTER.

Toolbar: Draw
Command entry: MTEXT

Quick Reference

Commands

DDEDIT

Edits single-line text, dimension text, attribute definitions, and feature control frames.

MLEADER

Creates a multileader object.

MTEXT

Creates a multiline text object.

SPELL

Checks spelling in a drawing.

STYLE

Creates, modifies, or specifies text styles.

TEXT

Creates a single-line text object.

System Variables

DIMASZ

Controls the size of dimension line and leader line arrowheads.

DIMLDRBLK

Specifies the arrow type for leaders.

Utilities

No entries

Command Modifiers

No entries

Create Text

You can create text using several methods, depending on your needs.

Overview of Creating Text

The text you add to your drawings conveys a variety of information. It may be a complex specification, title block information, a label, or even part of the drawing.

Single-Line Text

For short entries that do not require multiple fonts or lines, create single-line text. Single-line text is most convenient for labels.

Multiline Text

For long, complex entries, create multiline, or paragraph text. Multiline text consists of any number of text lines or paragraphs that fit within a width you specify; it can extend vertically to an indefinite length.

Regardless of the number of lines, each set of paragraphs created in a single editing session forms a single object, which you can move, rotate, erase, copy, mirror, or scale.

There are more editing options for multiline text than there are for single-line text. For example, you can apply underlining, fonts, color, and text height changes to individual characters, words, or phrases within a paragraph.

Annotative Text

Use text for notes and labels in your drawing. You create annotative text by using an annotative text style, which sets the height of the text on the paper.

For more information about creating and working with an annotative text, see Create Annotative Text on page 1404.

See also:

- Scale Annotations on page 1393

To create single-line text

1. Click Home tab ➤ Annotation panel ➤ Single Line Text. 
2. Specify the insertion point for the first character. If you press ENTER, the program locates the new text immediately below the last text object you created, if any.

3 Specify the height of the text. This prompt is displayed only if text height is set to 0 in the current text style.

   A rubber-band line is attached from the text insertion point to the cursor. Click to set the height of the text to the length of the rubber-band line.

4 Specify a text rotation angle.

   You can enter an angle value or use your pointing device.

5 Enter the text. At the end of each line, press ENTER. Enter more text as needed.

   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.

   If you specify another point during this command, the cursor moves to that point, and you can continue typing. Every time you press ENTER or specify a point, a new text object is created.

6 Press ENTER on a blank line to end the command.

Command entry: TEXT

To create multiline text

1 Click Home tab ➤ Annotation panel ➤ Multiline Text.

2 Specify opposite corners of a bounding box to define the width of the multiline text object.

   If the ribbon is active, the MTEXT ribbon contextual tab displays. If the ribbon is not active, the In-Place Text Editor is displayed.

3 To indent the first line of each paragraph, drag the first-line indent slider on the ruler. To indent the other lines of each paragraph, drag the paragraph slider.

4 To set tabs, click the ruler where you want a tab stop.

5 If you want to use a text style other than the default, on the ribbon, click the Annotate tab, Text panel. Select the desired text style from the drop-down list.

6 Enter text.

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.

7 To override the current text style, select text as follows:

- To select one or more letters, click and drag the pointing device over the characters.
- To select a word, double-click the word.
- To select a paragraph, triple-click the paragraph.

8 On the ribbon, make format changes as follows:

- To change the font of the selected text, select a font from the list.
- To change the height of the selected text, enter a new value in the Height box.

  NOTE The MText height value is reset to 0 if its default height is not modified during creation.

- To format text in a TrueType font with boldface or italics, or to create underlined or overlined text for any font, click the corresponding button on the ribbon. SHX fonts do *not* support boldface or italics.
- To apply color to selected text, choose a color from the Color list. Click Other to display the Select Color dialog box.

9 To save your changes and exit the editor, use one of the following methods:

- On the MTEXT ribbon contextual tab, in the Close panel, click Close Text Editor.
- Click in the drawing outside the editor.
- Press CTRL+ENTER.

Toolbar: Draw 
Command entry: MTEXT

Quick Reference

Commands

MLEADER

Creates a multileader object.

MTEXT

Creates a multiline text object.

QLEADER

Creates a leader and leader annotation.

TEXT

Creates a single-line text object.

System Variables

DIMASSOC

Controls the associativity of dimension objects and whether dimensions are exploded.

DIMASZ

Controls the size of dimension line and leader line arrowheads.

DIMLDRBLK

Specifies the arrow type for leaders.

Utilities

No entries

Command Modifiers

No entries

Create Single-Line Text

You can use single-line text to create one or more lines of text, where each text line is an independent object that you can relocate, reformat, or otherwise modify.

Use single-line text (TEXT) to create one or more lines of text, ending each line when you press ENTER. Each text line is an independent object that you can relocate, reformat, or otherwise modify.

When you create single-line text, you assign a text style and set alignment. The text style sets the default characteristics of the text object. The alignment determines what part of the text character aligns with the insertion point. Use the TEXT command to enter the text in-place, or enter **-text** at the Command prompt to enter the text at the Command prompt.

You can insert a field in single-line text. A field is text that is set up to display data that might change. When the field is updated, the latest value of the field is displayed.

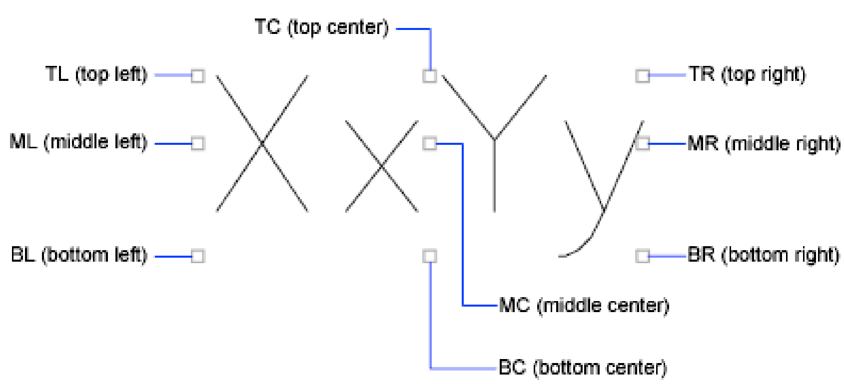
The text styles used for single-line text are the same as those used for multiline text. When you create text, you assign an existing style by entering its name at the Style prompt. If you need to apply formatting to individual words and characters, use multiline text instead of single-line text.

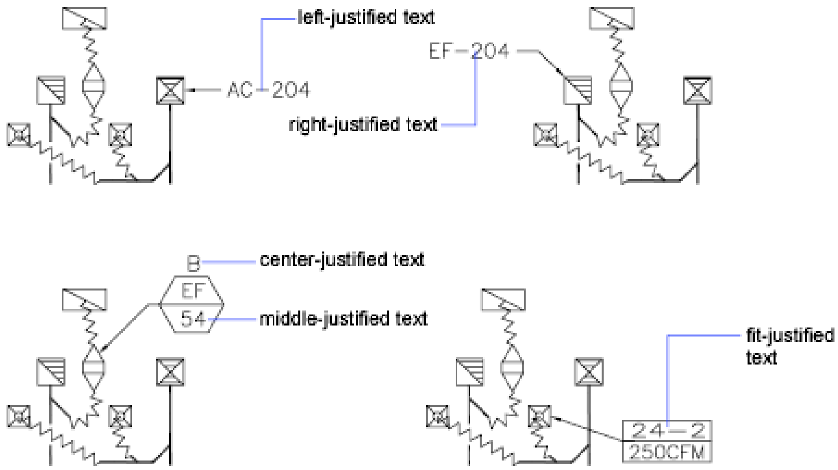
You can also compress single-line text to fit between points that you specify. This option stretches or squeezes the text to fill the designated space.

The DTEXTED system variable specifies the user interface displayed for editing single-line text.

Align Single-Line Text

As you create text, you can align it. That is, you can justify it with one of the alignment options shown in the following illustrations. Left alignment is the default. To left-align text, do not enter an option at the Justify prompt.





See also:

- Use Fields in Text on page 1532

To create single-line text

1 Click Home tab ➤ Annotation panel ➤ Single Line Text. 

2 Specify the insertion point for the first character. If you press ENTER, the program locates the new text immediately below the last text object you created, if any.

3 Specify the height of the text. This prompt is displayed only if text height is set to 0 in the current text style.

   A rubber-band line is attached from the text insertion point to the cursor. Click to set the height of the text to the length of the rubber-band line.

4 Specify a text rotation angle.

   You can enter an angle value or use your pointing device.

5 Enter the text. At the end of each line, press ENTER. Enter more text as needed.

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.

If you specify another point during this command, the cursor moves to that point, and you can continue typing. Every time you press ENTER or specify a point, a new text object is created.

6 Press ENTER on a blank line to end the command.

Command entry: TEXT

To specify a text style when you create single-line text

1 Click Home tab ➤ Annotation panel ➤ Single Line Text. 
2 Enter **s** (Style).
3 At the Style Name prompt, enter an existing text style name.
   If you first want to see a list of text styles, enter **?** and press ENTER twice.
4 Continue creating text.

Command entry: TEXT

To align single-line text as you create it

1 Click Home tab ➤ Annotation panel ➤ Single Line Text. 
2 Enter **j** (Justify).
3 Enter an alignment option. For example, enter **br** to align text at its bottom-right corner.
4 Continue creating text.

Command entry: TEXT

Quick Reference

Commands

QTEXT

Controls the display and plotting of text and attribute objects.

STYLE

Creates, modifies, or specifies text styles.

TEXT

Creates a single-line text object.

System Variables

DTEXTED

Specifies the user interface displayed for editing single-line text.

FONTALT

Specifies the alternate font to be used when the specified font file cannot be located.

FONTMAP

Specifies the font mapping file to be used.

MIRRTEXT

Controls how the MIRROR command reflects text.

QTEXTMODE

Controls how text is displayed.

TEXTEVAL

Controls how text strings entered with TEXT (using AutoLISP) or with -TEXT are evaluated.

TEXTFILL

Controls the filling of TrueType fonts while plotting and rendering.

TEXTQLTY

Sets the resolution tessellation fineness of text outlines.

TEXTSIZE

Sets the default height for new text objects drawn with the current text style.

TEXTSTYLE

Sets the name of the current text style.

Utilities

No entries

Command Modifiers

No entries

Create Multiline Text

A multiline text (mtext) object includes one or more paragraphs of text that can be manipulated as a single object.

Overview of Multiline Text

You can create a multiline text (mtext) object by entering or importing text.

You can create one or more paragraphs of multiline text (mtext) in the MTEXT ribbon contextual tab (if the ribbon is active), or the In-Place Text Editor (or an alternative text editor, if the ribbon is not active) You can also use Command prompts. You can insert text from a file saved in ASCII or RTF format.

Before entering or importing text, you specify opposite corners of a text bounding box that defines the width of the paragraphs in the multiline text object. The length of the multiline text object depends on the amount of text, not the length of the bounding box. You can use grips to move or rotate a multiline text object.

NOTE Multiline text objects and imported text files are limited to 256 KB in size.

The MTEXT ribbon contextual tab and In-Place Text Editor display the bounding box with a ruler at the top. If the ribbon is not active, the Text Formatting toolbar is also displayed. The editor is transparent so that, as you create text, you can see whether the text overlaps other objects. To turn off transparency while you work, select Opaque Background on the Options menu. You can also make the background of the finished multiline text object opaque and set its color.

You can also insert fields in multiline text. A field is text that is set up to display data that might change. When the field is updated, the latest value of the field is displayed.

Text Style

Most characteristics of the text are controlled by the text style, which sets the default font and other options, such as line spacing, justification, and color. You can use the current text style or select a new one. The STANDARD text style is the default.

Within the multiline text object, you can override the current text style by applying formatting such as underlining, boldface, and different fonts to individual characters. You can also create stacked text, such as fractions or geometric tolerances and insert special characters, including Unicode characters, for TrueType fonts.

NOTE Not all SHX and TrueType text fonts support Unicode characters.

Text Properties

In the Properties palette, you can view and change the object properties of a multiline text object, including properties that apply specifically to text.

- Justification determines where text is inserted with respect to the bounding box and sets the direction of text flow as text is entered.
- Line space options control the amount of space between lines of text.
- Width defines the width of the bounding box and therefore controls where the text wraps to a new line.
- Background inserts an opaque background so that objects under the text are masked.

To create multiline text

1. Click Home tab ➤ Annotation panel ➤ Multiline Text. 
2. Specify opposite corners of a bounding box to define the width of the multiline text object.

   If the ribbon is active, the MTEXT ribbon contextual tab displays. If the ribbon is not active, the In-Place Text Editor is displayed.
3. To indent the first line of each paragraph, drag the first-line indent slider on the ruler. To indent the other lines of each paragraph, drag the paragraph slider.
4. To set tabs, click the ruler where you want a tab stop.

5 If you want to use a text style other than the default, on the ribbon, click the Annotate tab, Text panel. Select the desired text style from the drop-down list.

6 Enter text.

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.

---

7 To override the current text style, select text as follows:

- To select one or more letters, click and drag the pointing device over the characters.
- To select a word, double-click the word.
- To select a paragraph, triple-click the paragraph.

8 On the ribbon, make format changes as follows:

- To change the font of the selected text, select a font from the list.
- To change the height of the selected text, enter a new value in the Height box.

  NOTE The MText height value is reset to 0 if its default height is not modified during creation.

- To format text in a TrueType font with boldface or italics, or to create underlined or overlined text for any font, click the corresponding button on the ribbon. SHX fonts do *not* support boldface or italics.
- To apply color to selected text, choose a color from the Color list. Click Other to display the Select Color dialog box.

9 To save your changes and exit the editor, use one of the following methods:

- On the MTEXT ribbon contextual tab, in the Close panel, click Close Text Editor.
- Click in the drawing outside the editor.
- Press CTRL+ENTER.

Toolbar: Draw
Command entry: MTEXT

To insert symbols or special characters in multiline text

1 If the ribbon is active, double-click a multiline text object to open the MTEXT ribbon contextual tab. If the ribbon is not active, the In-Place Text Editor is displayed.

2 On either the ribbon contextual tab or expanded toolbar, click Symbol.

3 Click one of the options on the menu, or click Other to display the Character Map dialog box.

   To access the Character Map dialog box, you must have *charmap.exe* installed. See Microsoft® Windows® Help for information about adding programs to your system.

4 In the Character Map dialog box, select a font.

5 Select a character, and use one of the following methods:

   - To insert a single character, drag the selected character into the editor.
   - To insert multiple characters, click Select to add each character to the Characters to Copy box. When you have all the characters you want, click Copy. Right-click in the editor. Click Paste.

6 To save your changes and exit the editor, use one of the following methods:

   - On the MTEXT ribbon contextual tab, in the Close panel, click Close Text Editor.
   - Click in the drawing outside the editor.
   - Press CTRL+ENTER.

Toolbar: Draw
Command entry: MTEXT

To add an opaque background or fill to a multiline text object

1. If the ribbon is active, double-click a multiline text object to open the MTEXT ribbon contextual tab. If the ribbon is not active, the In-Place Text Editor is displayed.
2. On the ribbon contextual tab, click Background Mask. In the editor, right-click. Click Background Mask.
3. In the Background Mask dialog box, select Use Background Mask.
4. Enter a value for Border Offset Factor.

   The value is based on the text height. A factor of 1.0 exactly fits the multiline text object. A factor of 1.5 (the default) extends the background by 0.5 times the text height.
5. Under Fill Color, do one of the following:
   - Select the Use Drawing Background Color option.
   - Select a color for the background, or click Select Color to open the Select Color dialog box.
6. Click OK to return to the editor.
7. To save your changes and exit the editor, use one of the following methods:
   - On the MTEXT ribbon contextual tab, in the Close panel, click Close Text Editor.
   - Click in the drawing outside the editor.
   - Press CTRL+ENTER.

   The opaque background is applied when you exit the editor.

Toolbar: Draw 
Command entry: MTEXT

To change the Windows font smoothing setting to improve visibility of text in the Multiline Text editor

1. On the Windows desktop, right-click. Click Properties.
2. In the Display Properties dialog box, click the Appearance tab.

3 Click Effects.

4 In the Effects dialog box, click the Use the Following Method to Smooth Edges of Screen Fonts to clear the setting.

5 Click OK to exit the Effects dialog box.

6 Click OK to exit the Display Properties dialog box.

Quick Reference

Commands

MTEXT

Creates a multiline text object.

QTEXT

Controls the display and plotting of text and attribute objects.

STYLE

Creates, modifies, or specifies text styles.

System Variables

MTEXTCOLUMN

Sets the default column setting for an mtext object.

MTEXTED

Sets the application for editing multiline text objects.

MTEXTFIXED

Sets the display size and orientation of multiline text in a specified text editor.

MTEXTTOOLBAR

Controls the display of the Text Formatting toolbar.

MTJIGSTRING

Sets the content of the sample text displayed at the cursor location when the MTEXT command is started.

QTEXTMODE

Controls how text is displayed.

TEXTFILL

Controls the filling of TrueType fonts while plotting and rendering.

TEXTQLTY

Sets the resolution tessellation fineness of text outlines.

TEXTSIZE

Sets the default height for new text objects drawn with the current text style.

TEXTSTYLE

Sets the name of the current text style.

Utilities

No entries

Command Modifiers

No entries

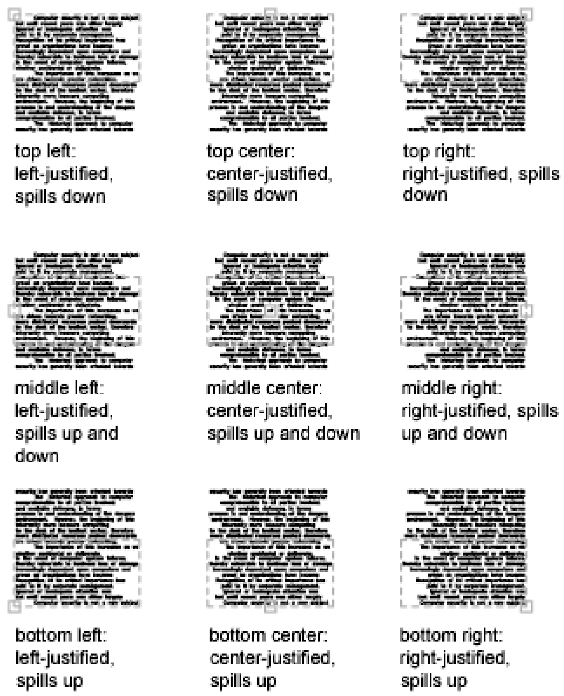
Justify Multiline Text

Justification of multiline text objects controls both text alignment and text flow relative to the text insertion point.

Justification controls both text alignment and text flow relative to the text insertion point. Text is left-justified and right-justified with respect to the boundary rectangle that defines the text width. Text flows from the insertion point, which can be at the middle, the top, or the bottom of the resulting text object.

There are nine justification settings for multiline text.

If a single word is longer than the width of the paragraph, the word will extend beyond the paragraph boundary.



To justify multiline text

1 Click View tab ➤ Palettes panel ➤ Properties. 

2 Select the multiline text object.

3 On the Properties palette, select one of the Justification options.

4 Click outside the Properties palette.

Command entry: PROPERTIES

Quick Reference

Commands

PROPERTIES

Controls properties of existing objects.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Format Characters Within Multiline Text

You can override the text style and apply different formatting to individual words and characters within multiline text.

The format changes affect only the text you select; the current text style is not changed.

You can specify a different font and text height and apply boldface, italics, underlining, overlining, and color. You can also set an obliquing angle, change the space between characters, and make characters wider or narrower. The Remove Formatting option on the menu of options resets the character attributes of selected text to the current text style and also resets the text color to the color of the mtext object.

The text height setting specifies the height of capitalized text. For more information about how height is calculated, see MTEXT.

See also:

- Use an Alternate Text Editor on page 1576

To format characters in multiline text

1. If the ribbon is active, double-click a multiline text object to open the MTEXT ribbon contextual tab. If the ribbon is not active, the In-Place Text Editor is displayed.
2. Select the text you want to format:
   - To select one or more letters, click and drag the pointing device over the characters.
   - To select a word, double-click the word.

- To select a paragraph, triple-click the paragraph.

3 On either the ribbon contextual tab or toolbar, make format changes as follows:
   - To change the font of the selected text, select a font from the list.
   - To change the height of the selected text, enter a new value in the Height box.
   - To format text in a TrueType font with boldface or italics, or to create underlined or overlined text for any font, click the corresponding button on the ribbon. SHX fonts do *not* support boldface or italics.
   - To apply color to selected text, select a color from the Color list. Click Other to display the Select Color dialog box.
   - To set an obliquing angle for the text, enter a value between -85 and 85. A positive value slants text to the right. A negative value slants text to the left.
   - To change letter spacing in the selected text, enter a new value.
   - To change the width of characters in the selected text, enter a new value.

4 To save your changes and exit the editor, use one of the following methods:
   - On the MTEXT ribbon contextual tab, in the Close panel, click Close Text Editor.
   - Click in the drawing outside the editor.
   - Press CTRL+ENTER.

Toolbar: Draw
Command entry: MTEXT

Quick Reference

Commands

DDEDIT

Edits single-line text, dimension text, attribute definitions, and feature control frames.

MTEXT

Creates a multiline text object.

PROPERTIES

Controls properties of existing objects.

STYLE

Creates, modifies, or specifies text styles.

System Variables

TEXTFILL

Controls the filling of TrueType fonts while plotting and rendering.

TEXTSTYLE

Sets the name of the current text style.

Utilities

No entries

Command Modifiers

No entries

Create Lists in Multiline Text

You can create bulleted lists, lettered or numbered lists, or simple outlines in multiline text.

Lines of multiline text can be formatted as a list. When you add or delete an item, or move an item up or down a level, the list numbering automatically adjusts. You can remove and reapply list formatting with the same method as used in most text editors.

Use Automatic List Formatting

By default, list formatting is applied to all text that looks like a list. Text that meets all the following criteria is considered to be a list:

- The line begins with one or more letters or numbers or a symbol.
- The letters or numbers is followed by punctuation.
- A space after the punctuation is created by pressing TAB.
- The text following the space is ended by ENTER or SHIFT+ENTER.

NOTE If you do not want list formatting applied to all text that fits the criteria, clear the Allow Bullets and Lists option. (Right-click in the In-Place Text Editor, click Bullets and Lists, and clear Allow Bullets and Lists.) When Allow Bullets and Lists is not checked, you cannot create new formatted lists in the multiline text object.

To create a list, use one of the following methods:

- Apply list formatting to new or selected text.
- Use Auto-list (on by default) and type the elements of a list.
- With Auto-list off, type the elements of a list and close and reopen the editor to convert the text to a list.

Apply List Formatting

When you apply list formatting, you can specify bullets, uppercase or lowercase letters, or numbers. Default settings are used for the type of list you choose. Letters or numbers are followed by a period. Nested lists use a double bullet, letter, or number. Items are indented based on the tab stops on the ruler in the In-Place Text Editor.

Use Auto-list to Type a List

When Auto-list is on, you can create a list as you type. You can use letters, numbers, or symbols.

For example, in the editor, enter \U+25CB, press TAB, and then enter some text. This creates a empty circle style bullet.

Not all symbols are available from the character map for a particular text font. However, if you specify the Unicode text directly (\U+25CB in this case), you can always get the bullet format of your choice.

NOTE Press TAB after you enter the Unicode text or symbol, or it will remain a separate character.

You can also paste a symbol from the Character Map dialog box

The following characters can be used as punctuation after the number or letter when you type a list but cannot be used as bullets:

Character	Description
.	Period
:	Colon
)	Close parenthesis
>	Close angle bracket
]	Close square bracket
}	Close curly bracket

Paste a List from Another Document

If you copy a nested bulleted list (a list within a list) from Microsoft Word and paste the list into multiline text, the bullets that are displayed as empty circles cannot be formatted like other bullets in multiline text. This is because Word uses the letter *o* instead of a bullet for nested bulleted lists. You can remove formatting from the nested list and reapply to change the bullets to double bullets.

To format multiline text as a list

1. Click Home tab ➤ Annotation panel ➤ Multiline Text. 
2. Specify opposite corners of a bounding box to define the width of the multiline text object.
3. To expand the Text Formatting toolbar, click Options button ➤ Show Options.
4. If you are converting multiline text to a list, select the paragraphs.

NOTE List formatting is only available when the Allow Bullets and Lists option is checked (the default).

5 On the expanded toolbar, click Numbering, Bullets, or Uppercase Letters.
   - **Numbering**. Uses numbers with periods for the items in a list.
   - **Bullets**. Uses a bullet or other character for the items in a list.
   - **Uppercase Letters**. Uses uppercase letters with periods for the items in a list. If the list has more items than the alphabet has letters, the sequence continues by using double letters. To use lowercase letters, right-click in the editor. Click Bullets and Lists ➤ Lettered ➤ Lowercase.

6 If you are creating new list items, enter the text.

7 To end the list, press ENTER to move to a new line. Click the button that you clicked to start the list.

8 To save your changes and exit the editor, use one of the following methods:
   - On the MTEXT ribbon contextual tab, in the Close panel, click Close Text Editor.
   - Click in the drawing outside the editor.
   - Press CTRL+ENTER.

Toolbar: Draw 
Command entry: MTEXT

To remove list formatting from multiline text

1 If the ribbon is active, double-click a multiline text object to open the MTEXT ribbon contextual tab. If the ribbon is not active, the In-Place Text Editor is displayed.

2 Select the list items.

3 To expand the Text Formatting toolbar, click Options button ➤ Show Options.

4 On the expanded toolbar, click the active list button to make it inactive: Numbering, Bullets, or Uppercase Letters.

NOTE If the list uses lowercase letters, click Uppercase Letters to convert the list to uppercase. Then click Uppercase Letters to make it inactive.

---

5 To save your changes and exit the editor, use one of the following methods:
   - On the MTEXT ribbon contextual tab, in the Close panel, click Close Text Editor.
   - Click in the drawing outside the editor.
   - Press CTRL+ENTER.

Toolbar: Draw

Command entry: MTEXT

To create a lettered or numbered list in multiline text as you type

1 Click Home tab ➤ Annotation panel ➤ Multiline Text.

2 Specify opposite corners of a bounding box to define the width of the multiline text object.

3 To expand the Text Formatting toolbar, click Options button ➤ Show Options.

4 Click Bullets and Lists. Verify that Allow Auto-list and Allow Bullets and Lists are checked.

5 Enter a letter or a number and a period (or other punctuation).

   The following characters can be used as punctuation after letters and numbers: period (.), colon (:), close parenthesis ()), close angle bracket (>), close square bracket (]), and close curly bracket (}).

6 Press TAB.

7 Enter the text of the list item. Press ENTER to move to the next item, or press SHIFT+ENTER to add a plain paragraph before the next item.

   The item is automatically lettered or numbered in sequence.

8 Press ENTER twice to end the list.

9 To save your changes and exit the editor, use one of the following methods:

- Click in the drawing outside the editor.
- Press CTRL+ENTER.
- Click the Close Editor icon in the Multiline Text panel.

Toolbar: Draw
Command entry: MTEXT

To create a bulleted list in multiline text as you type

1 Click Home tab ➤ Annotation panel ➤ Multiline Text.

2 Specify opposite corners of a bounding box to define the width of the multiline text object.

3 In the Options panel, click the Options icon.

4 Click Bullets and Lists. Verify that Allow Auto-list and Allow Bullets and Lists are selected (tick mark).

5 Start a line of text by entering **\U+2022** (the Unicode string for a bullet) or by selecting a bullet character or another symbol.

NOTE The following characters cannot be used as bullets: period (.), colon (:), close parenthesis ()), close angle bracket (>), close square bracket (]), and close curly bracket (}).

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6 Alternatively, click Options ➤ Symbol ➤ Other.
The Character Map dialog box appears.

7 Double-click a symbol to copy it to the clipboard.

8 Close the Character Map dialog box.

9 Paste the symbol to the drawing, and press the up-arrow and End key to return the cursor to the same line.

10 Press TAB.

11 Enter the text of the list item. Press ENTER to move to the next item, or press SHIFT+ENTER to add a plain paragraph before the next item.

The bullet character is automatically added to the next item.

12 Press ENTER twice to end the list.

13 To save your changes and exit the editor, use one of the following methods:

- On the MTEXT ribbon contextual tab, in the Close panel, click Close Text Editor.
- Click in the drawing outside the editor.
- Press CTRL+ENTER.

Toolbar: Draw A
Command entry: MTEXT

To move a list item in multiline text down a level

1 If the ribbon is active, double-click a multiline text object to open the MTEXT ribbon contextual tab. If the ribbon is not active, the In-Place Text Editor is displayed.

2 Place the cursor at the beginning of a list item and press TAB.

The item moves down one level and begins a nested list.

3 Press ENTER to start the next item at the same level, or press SHIFT+TAB to move the item up a level.

4 To save your changes and exit the editor, use one of the following methods:

- On the MTEXT ribbon contextual tab, in the Close panel, click Close Text Editor.
- Click in the drawing outside the editor.
- Press CTRL+ENTER.

Toolbar: Draw A

Command entry: MTEXT

To separate an existing list

1 If the ribbon is active, double-click a multiline text object to open the MTEXT ribbon contextual tab. If the ribbon is not active, the In-Place Text Editor is displayed.

2 Select a sequence of list items or place the cursor at the beginning of the item where you want to start the new list.

3 Right-click in the editor. Click Bullets and Lists ➤ Restart.

   The selected items are renumbered as a separate sequence. If you select items in the middle of a list, unselected items below the selected items also become part of the new list.

4 To continue the original list below the new list, select the first item below the new list.

5 Right-click in the editor. Click Bullets and Lists ➤ Continue.

   The selected item and the items following it are renumbered to continue the previous list.

6 To save your changes and exit the editor, use one of the following methods:

   - Click OK on the toolbar.
   - Click in the drawing outside the editor.
   - Press CTRL+ENTER.

Toolbar: Draw A
Command entry: MTEXT

To convert the lists in a multiline text object to plain text

1 If the ribbon is active, double-click a multiline text object to open the MTEXT ribbon contextual tab. If the ribbon is not active, the In-Place Text Editor is displayed.

2 Right-click in the editor. Click Bullets and Lists. Remove the check mark next to Allow Bullets and Lists.

The plain text lists retain their bullets, numbers, or letters. If you add an item to the list, the number or letter sequence does not change.

Quick Reference

Commands

DDEDIT

Edits single-line text, dimension text, attribute definitions, and feature control frames.

MTEXT

Creates a multiline text object.

PROPERTIES

Controls properties of existing objects.

System Variables

TEXTFILL

Controls the filling of TrueType fonts while plotting and rendering.

TEXTQLTY

Sets the resolution tessellation fineness of text outlines.

Utilities

No entries

Command Modifiers

No entries

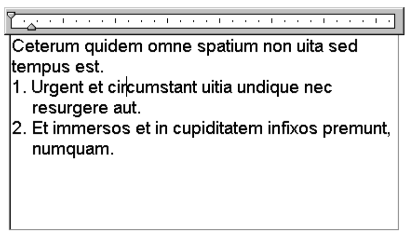
Indent Multiline Text and Use Tabs

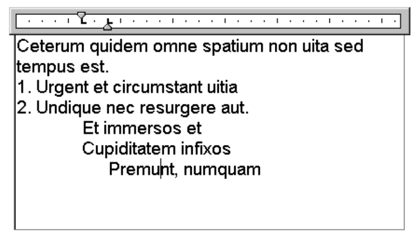
You can control how paragraphs are indented in a multiline text (mtext) object. The ruler in the In-Place Text Editor shows the settings for the current paragraph.

Tabs and indents that you set before you start to enter text apply to the whole multiline text object. To apply different tabs and indents to individual paragraphs, click in a single paragraph or select multiple paragraphs and then change the settings.

Sliders on the ruler show indentation relative to the left side of the bounding box. The top slider indents the first line of the paragraph, and the bottom slider indents the other lines of the paragraph.

The long tick marks on the ruler show the default tab stops. If you click the ruler to set your own tabs, the ruler displays a small, L-shaped marker at each custom tab stop. You can delete a custom tab stop by dragging the marker off the ruler.





To create paragraphs with hanging indentation

1. If the ribbon is active, double-click a multiline text object to open the MTEXT ribbon contextual tab. If the ribbon is not active, the In-Place Text Editor is displayed.
2. Select the paragraphs you want to indent.
3. On the ruler, slide the top indent marker to where you want the first line of the paragraphs to start.

   NOTE The ruler displays the tabs and indents that are set for the selected paragraphs or, if no text is selected, the paragraph where the cursor is located. The default tab stops are the long tick marks on the ruler. To set a custom tab stop, click the ruler where you want the tab stop.

   ---
4. Slide the bottom indent marker to where you want the other lines of the paragraphs to start.

   This step indents turnover lines in paragraphs that are more than one line long.
5. To change the indentation, select the paragraphs you want to change, click the ruler to set new tab stops, if needed, and move the indent markers.

6 To save your changes and exit the editor, use one of the following methods:
   - On the MTEXT ribbon contextual tab, in the Close panel, click Close Text Editor.
   - Click in the drawing outside the editor.
   - Press CTRL+ENTER.

Toolbar: Draw A
Command entry: MTEXT

Quick Reference

Commands

DDEDIT

Edits single-line text, dimension text, attribute definitions, and feature control frames.

MTEXT

Creates a multiline text object.

PROPERTIES

Controls properties of existing objects.

System Variables

TEXTFILL

Controls the filling of TrueType fonts while plotting and rendering.

TEXTQLTY

Sets the resolution tessellation fineness of text outlines.

Utilities

No entries

Command Modifiers

No entries

Specify the Line Spacing Within Multiline Text

Line spacing for multiline text is the distance between the baseline (bottom) of one line of text and the baseline of the next line of text. The line space factor applies to the entire multiline text object, not to selected lines.

You can set the spacing increment to a multiple of single line spacing, or as an absolute distance. Single spacing is 1.66 times the height of the text characters.

The default line space style, At Least, automatically increases line spacing to accommodate characters that are too large to fit the line spacing you set for the multiline text object. Use the other line space style, Exactly, to line up text in tables.

To ensure that line spacing is identical in multiple multiline text objects, use Exactly and set the Line Space Factor to the same value in each multiline text object.

NOTE Using Exactly can cause text in lines located above or below lines with large font characters to overlap the larger characters.

To change the line spacing of multiline text

1 Click View tab ➤ Palettes panel ➤ Properties. 

2 Select the multiline text object you want to edit.

3 In the Properties palette, for Line Space Style, select one of the following:

   - **At Least.** Adjusts lines of text automatically based on the height of the largest character in the line. More space is added between lines of text with taller characters. This is the default setting.

   - **Exactly.** Forces the line spacing to be the same size for all lines of text regardless of format differences such as font or text height.

4 Change the line spacing by entering a new value for either of the following options. The two line spacing options provide different ways to set the same thing:
   - **Line Space Factor.** Sets the line spacing to a multiple of single-line spacing. Single spacing is 1.66 times the height of the text characters.
   - **Line Space Distance.** Sets the line spacing to an absolute value measured in drawing units. Valid values must be between 0.0833 and 1.3333.

NOTE After you exit the Properties palette, the value of the *other* line spacing option is updated to correspond with the line spacing value that you entered.

Toolbar: Standard 
Command entry: PROPERTIES

Quick Reference

Commands

PROPERTIES

Controls properties of existing objects.

System Variables

TSPACEFAC

Controls the multiline text line-spacing distance measured as a factor of text height.

TSPACETYPE

Controls the type of line spacing used in multiline text.

Utilities

No entries

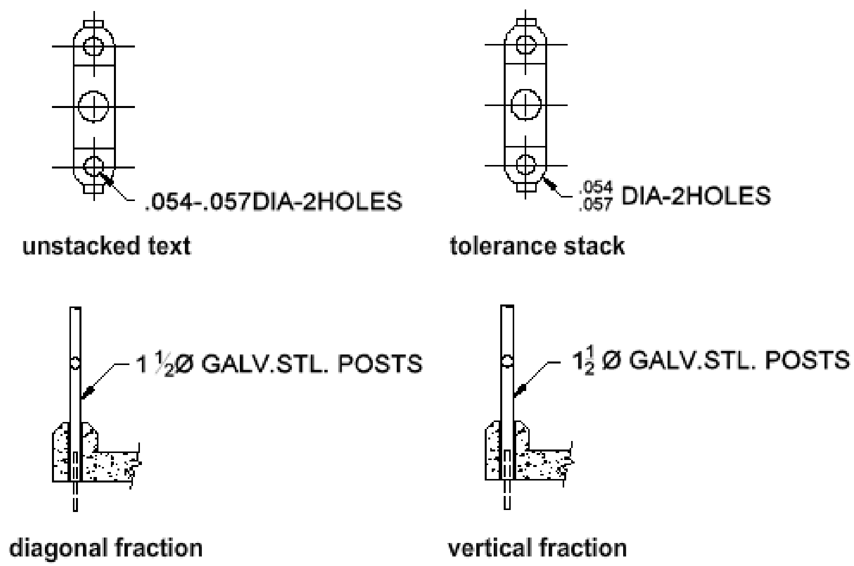
Command Modifiers

No entries

Create Stacked Characters Within Multiline Text

Characters representing fractions and tolerances can be formatted to conform to several standards.

Stacked text refers to the fraction and tolerance formats applied to characters within multiline text object and multileaders.



You use special characters to indicate how selected text should be stacked.

- Slash (/) stacks text vertically, separated by a horizontal line.
- Pound sign (#) stacks text diagonally, separated by a diagonal line.
- Carat (^) creates a tolerance stack, which is stacked vertically and not separated by a line.

To stack characters manually within the In-Place Text Editor, select the text to be formatted, including the special stacking character, and click the Stack button on the Text Formatting toolbar.

Stack Numeric and Tolerance Characters Automatically

You can specify that numeric characters entered before and after a slash, pound sign, or carat will stack automatically. For example, if you enter **1#3** followed by a nonnumeric character or space, the AutoStack Properties dialog box is displayed by default, and you can change the settings in the AutoStack dialog box to specify your formatting preferences.

The automatic stacking feature applies only to numeric characters immediately before and after the slash, pound sign, and carat. For tolerance stacking, the +, -, and decimal character also stack automatically.

See also:

- Use an Alternate Text Editor on page 1576

To create stacked text

1. Click Home tab ➤ Annotation panel ➤ Multiline Text. 
2. Specify opposite corners of a bounding box to define the width of the multiline text object.
3. In either the MTEXT ribbon contextual tab or In-Place Text Editor, set text style and other multiline text properties as needed.
4. Enter the text you want to stack separated by one of the following characters:
   - Slash (/) stacks text vertically, separated by a horizontal line.
   - Pound sign (#) stacks text diagonally, separated by a diagonal line.
   - Carat (^) creates a tolerance stack, which is not separated by a line.

   If you enter numbers separated by stack characters and then enter a nonnumeric character or press SPACEBAR, the AutoStack Properties dialog box is displayed.
5. In the AutoStack Properties dialog box, you can choose to automatically stack numbers (not nonnumeric text) and to remove leading blanks. You can also specify whether the slash character creates a diagonal fraction

or creates a vertical fraction. If you do not want to use AutoStack, click Cancel to exit the dialog box.

6 Select the text that you want to stack, and click the Stack button on the toolbar.

7 To save your changes and exit the editor, use one of the following methods:
   - Click OK on the toolbar.
   - Click in the drawing outside the editor.
   - Press CTRL+ENTER.

Toolbar: Draw 
Command entry: MTEXT

To change stack properties

1 Double-click the multiline text object you want to edit.

2 In either the MTEXT ribbon contextual tab or In-Place Text Editor, select the stacked text.

3 Right-click in the editor. Click Properties.

4 In the Stack Properties dialog box, change settings as needed.

5 To set properties for automatic stacking, click Autostack.

6 To save your changes and exit the editor, use one of the following methods:
   - Click OK on the toolbar.
   - Click in the drawing outside the editor.
   - Press CTRL+ENTER.

Toolbar: Draw 
Command entry: MTEXT

To unstack text

1. Double-click the multiline text object you want to edit.
2. In either the MTEXT ribbon contextual tab or In-Place Text Editor, select the stacked text.
3. Click Stack on the Text Formatting toolbar.
4. To save your changes and exit the editor, use one of the following methods:
   - Click OK on the toolbar.
   - Click in the drawing outside the editor.
   - Press CTRL+ENTER.

Toolbar: Draw 
Command entry: MTEXT

Quick Reference

Commands

MTEXT

Creates a multiline text object.

System Variables

TSTACKALIGN

Controls the vertical alignment of stacked text.

TSTACKSIZE

Controls the percentage of stacked text fraction height relative to selected text's current height.

Utilities

No entries

Command Modifiers

No entries

Create and Edit Columns in Multiline Text

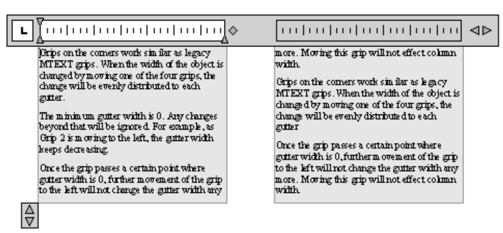
You can create and edit multiple columns using the In-Place Text Editor column options and column grips.

Multiple columns can be created and edited in either the MTEXT ribbon contextual tab or In-Place Text Editor and through the grip editing mode. Editing columns using grips allows you the flexibility of seeing the changes as you make them.

Columns follow a few rules. All columns have equal width and equal gutters. A gutter is the space between columns. The height of columns remains constant unless more text than the column can accommodate is added, or you manually move the editing grip to adjust the column height.

Editing Columns in the In-Place Text Editor

When you are working with columns using either the MTEXT ribbon contextual tab or In-Place Text Editor, the columns will be in a frame. If the Opaque background is turned on, the background covers each column, leaving gutter space blank. The ruler bar when applied, spans across all the columns, but the ruler is only active for the column that is selected as current.



Adding text to a column with an arbitrary height will not increase the column height even if text is already filling the column. Text will flow into another column.

You can also insert a column break to force text to start flowing into the next column. Anytime a column break is inserted, it is assumed that the current height of the column is fixed. To delete the break, highlight it and delete or or use the Backspace key right after the break.

Editing Columns in the Property Palette

You will be able to select Static or Dynamic columns, turn off columns and change column and gutter width through the Property Palette. Changing column width in the palette will exhibit results similar to changing width using grips. The palette is the only place that you can also change gutter setting.

To create multiple columns in the In-Place Text Editor

1 If the ribbon is active, double-click a multiline text object to open the MTEXT ribbon contextual tab. If the ribbon is not active, the In-Place Text Editor is displayed.

2 In the In-Place Text Editor, select a column option and suboption from columns list. You have a choice between Dynamic or Static columns.

   You have two suboptions with Dynamic Columns – Auto height or Manual height. Static Columns allows you to choose the number of columns.

3 Adjust the column height by moving the arrows located on the bottom left of the first column.

   NOTE The arrows on the ruler on the upper right side only adjust gutter width, not column width.

Toolbar: Draw A
Command entry: MTEXT

To adjust columns using grips

1 Select an area outside the mtext object. The In-Place Text Editor toolbar will disappear.

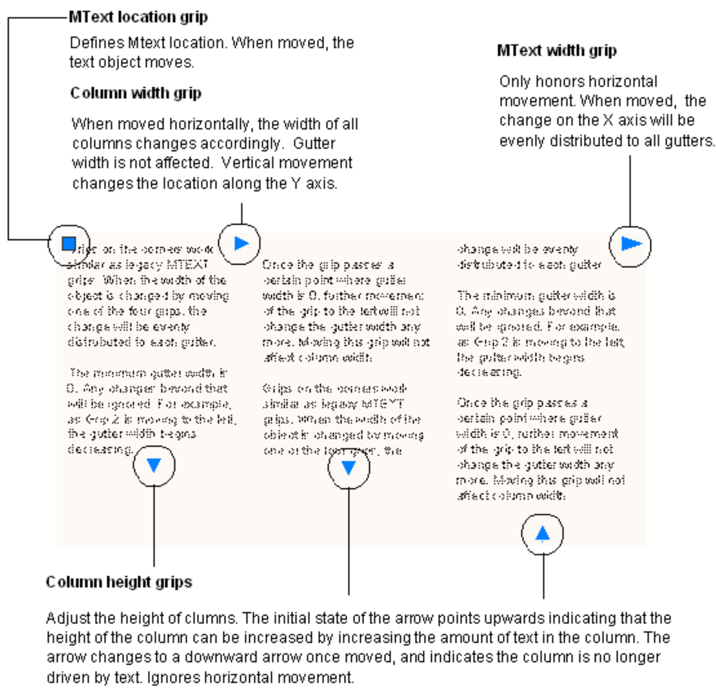
2 Click once in the text area and grips will appear.

   Grips control the location of the mtext object, the gutter width, and vertical and horizontal movement of colmns.

Toolbar: Draw A

Command entry: MTEXT

The following illustration demonstrates how grips are used with Dynamic Columns - Manual Height.



The following illustration demonstrates how grips are used with Static Columns.

NOTE In general, grips only update the mtext object after the mouse button is released.

Quick Reference

Commands

MTEXT

Creates a multiline text object.

System Variables

MTEXTCOLUMN

Sets the default column setting for an mtext object.

Utilities

No entries

Command Modifiers

No entries

Import Text from External Files

You can insert TXT or RTF text files into your drawing by either importing the text or dragging a file icon from Windows Explorer.

You can insert TXT or RTF text files created in word processors into your drawing by either importing the text or dragging a file icon from Windows Explorer.

Importing TXT or RTF files from other sources gives you the most flexibility. For example, you can create a text file of standard notes that you include in drawings. The imported text becomes a multiline text object, which you can edit and reformat. Text imported from a TXT file inherits the current text style. Text imported from an RTF file inherits the current text style *name*, but retains its original fonts and format. Imported text files are limited to 256 KB and must have a file extension of *.txt* or *.rtf*.

If you drag a text file into a drawing, text width is determined by line breaks and carriage returns in the original document. When you drag an RTF file into a drawing, the text is inserted as an OLE object.

If you use the Clipboard to paste text from another application, the text becomes an OLE object. If you use the Clipboard to paste text from another file, the text is inserted as a block reference, and it retains its original text style.

See also:

- Import OLE Objects

To import text files

1. Click Home tab ➤ Annotation panel ➤ Multiline Text. 
2. Specify opposite corners of a bounding box to define the width of the multiline text object.

3 Right-click in the editor. Click Import Text.

   The size limit for an imported file is 256 KB.

4 In the Select File dialog box, double-click the file you want to import, or select the file. Click Open.

   The text is inserted at the cursor location.

5 Change the text as needed.

6 To save your changes and exit the editor, use one of the following methods:

   - Click OK on the toolbar.
   - Click in the drawing outside the editor.
   - Press Ctrl+Enter.

Toolbar: Draw A
Command entry: MTEXT

To insert a text file using the drag-and-drop method

1 Open Windows Explorer, but make sure it *does not* fill the screen.

2 Display the folder that contains the TXT or RTF file you want.

3 Drag the TXT or RTF file icon onto the drawing. TXT files are inserted as multiline text objects using the current text style. RTF files are inserted as OLE objects.

Quick Reference

Commands

MTEXT

Creates a multiline text object.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Create Leaders

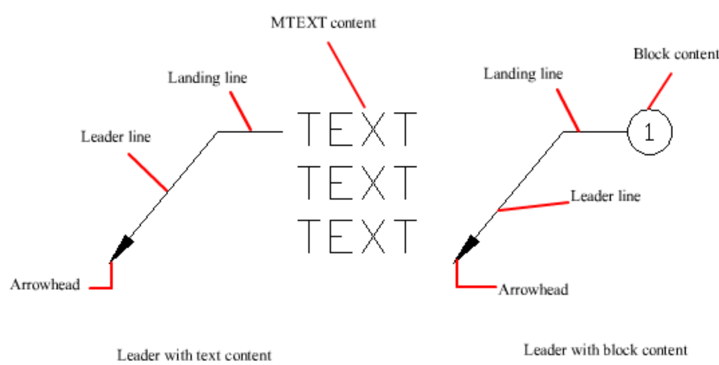
You can create, modify and add content to a leader object.

Overview of Leader Objects

A leader object is a line or a spline with an arrowhead at one end and a multiline text object or block at the other.

In some cases, a short horizontal line, called a landing, connects text or blocks and feature control frames to the leader line.

The landing and leader line are associated with the multiline text object or block, so when the landing is relocated, the content and leader line move along with it.



When associative dimensioning is turned on and object snaps are used to locate the leader arrowhead, the leader is associated with the object to which the arrowhead is attached. If the object is relocated, the arrowhead is relocated, and the landing stretches accordingly.

NOTE The leader object should not be confused with the leader line that is automatically generated as part of a dimension line.

To create a leader with straight lines

1 Click Home tab ➤ Annotation panel ➤ Multileader.

2 At the Command prompt, enter **o** to select options.

3 Enter **l** to specify leaders.

4 Enter **t** to specify the leader type.

5 Enter **s** to specify straight leaders.

6 In the drawing, click a start point for the leader head.

7 Click an end point for the leader.

8 Enter your MTEXT content.

9 On the Text Formatting toolbar, click OK.

Toolbar: Multileader
Command entry: MLEADER

Quick Reference

Commands

LEADER

Creates a line that connects annotation to a feature.

MLEADER

Creates a multileader object.

QLEADER

Creates a leader and leader annotation.

System Variables

DIMASSOC

Controls the associativity of dimension objects and whether dimensions are exploded.

DIMGAP

Sets the distance around the dimension text when the dimension line breaks to accommodate dimension text.

DIMLDRBLK

Specifies the arrow type for leaders.

MLEADERSCALE

Sets the overall scale factor applied to multileader objects.

Utilities

No entries

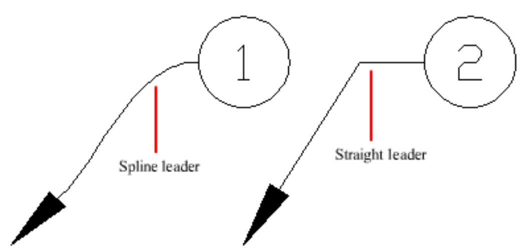
Command Modifiers

No entries

Create and Modify Leaders

A leader object typically consists of an arrowhead, an optional horizontal landing, a leader line or curve, and either a multiline text object or block.

You can create a leader line from any point or feature in a drawing and control its appearance as you draw. Leaders can be straight line segments or smooth spline curves.

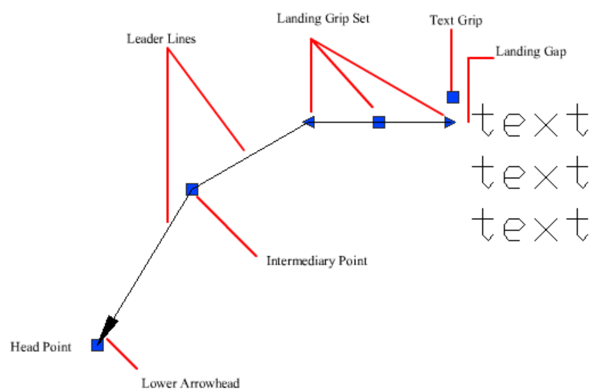


A multileader object, or MLEADER, comprises a leader and a note. It can be created arrowhead first, tail first, or content first. If a multileader style has been used, then the multileader can be created from that style

Multileader objects can contain multiple leader lines, each of which can have one or more segments, so that one note can point to multiple objects in your drawing. You can modify the properties of leader segment in the Properties palette. Using the MLEADEREDIT command, you can add leaders to, or remove leaders from, an established multileader object.

Annotative multileaders containing multiple leader segments can have different head points in each scale representation. Horizontal landings and arrowheads can have different sizes, and landing gaps can have different distances, depending on the scale representation. The appearance of the horizontal landing within a multileader, as well as the type of leader line (straight or spline) and number of leader segments will remain the same in all scale representations. For more information, see Create Annotative Leaders and Multileaders on page 1413.

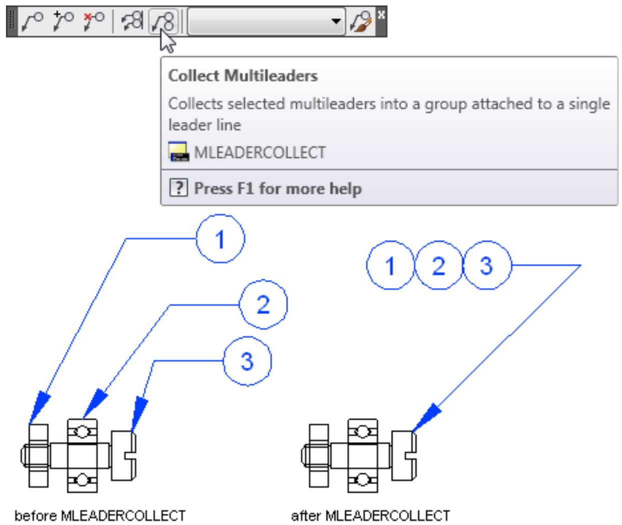
You can use grips to modify the look of a multileader. Using grips, you can lengthen or shorten a landing or leader line, or move the entire leader object.



Arrange Leaders

Multileaders can be arranged to add order and consistency to your drawing.

Multileader objects with blocks as content can be collected and attached to one landing line. Using the MLEADERCOLLECT command, multileaders can be collected horizontally, vertically, or within a specified area depending on your drawing needs.



Multileader objects can be sorted evenly along a specified line. Using the MLEADERALIGN command, selected multileaders can be aligned and evenly spaced as specified.

Associate Leaders with Objects

When associative dimensioning is turned on (DIMASSOC), the leader arrowhead can be associated with a location on an object using an object snap. If the object is relocated, the arrowhead remains attached to the object and the leader line stretches, but the multiline text remains in place.

See also:

- Create Annotative Leaders and Multileaders on page 1413

To create a leader with straight lines

1. Click Home tab ➤ Annotation panel ➤ Multileader. 
2. At the Command prompt, enter **o** to select options.
3. Enter **l** to specify leaders.
4. Enter **t** to specify the leader type.
5. Enter **s** to specify straight leaders.

6 In the drawing, click a start point for the leader head.

7 Click an end point for the leader.

8 Enter your MTEXT content.

9 On the Text Formatting toolbar, click OK.

Toolbar: Multileader
Command entry: MLEADER

To create a leader attached to block content at an angle

1 Click Home tab ➤ Annotation panel ➤ Multileader Style.

2 In the Multileader Style Manager, click New.

3 In the Create New Multileader Style dialog box, specify a name for the new multileader style.

4 In the Modify Multileader Style dialog box, Leader Structure tab, under Landing Settings, uncheck Automatically Include Landing.

5 On the Content tab, next to Multileader Type, choose Block content.

6 Under Block Options, next to Attachment, choose one of the following:

- Center Extents: Attaches the leader line to the center extent of the block content
- Insertion Point: Attaches the leader line to the block content from any point you specify

7 Click OK.

8 In the Multileader Style Manager, click Close.

9 Do one of the following:

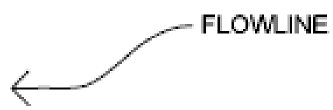
- Create a multileader object on page 1518
- Apply the new multileader style to an existing multileader object on page 1527

Menu: Format ➤ Multileader Style

Command entry: MLEADERSTYLE

To create a spline leader with text or a block

1. Click Home tab ➤ Annotation panel ➤ Multileader.
2. At the Command prompt, enter **o** to select options.
3. Enter **l** to specify leaders.
4. Enter **t** to specify the leader type.
5. Enter **p** to specify a spline leader.
6. In the drawing, click a start point for the leader head.
7. Click the end point for the leader.
8. Enter your MTEXT content.
9. In the Text Formatting toolbar, click OK.



Toolbar: Multileader
Command entry: MLEADER

To edit leader text

1. Double-click the text you want to edit.

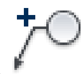
   If the ribbon is active, the MTEXT ribbon contextual tab is displayed. IF the ribbon is not active, the In-Place Text Editor is displayed for both single-line text and multiline text. The Text Formatting toolbar is not available for single-line text.

2. Edit the text.

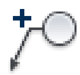
Command entry: DDEDIT

To create multiple leaders from the same annotation

1 Select the multileader.

2 Click Annotate tab ➤ Multileaders panel ➤ Add Leader. 

3 Specify the endpoint for the new leader.

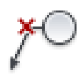
Toolbar: Multileader 
Command entry: MLEADEREDIT

To remove leaders from an annotation

1 Select the multileader.

2 On the Multileader toolbar, click Remove Leader.

3 Select the leader or leaders you want to remove. Press ENTER.

Toolbar: Multileader 
Command entry: MLEADEREDIT

To align and space leaders

1 Click Annotate tab ➤ Multileaders panel ➤ Align. 

2 Select the multileaders to be aligned. Press ENTER.

3 Specify a starting point in the drawing to begin the alignment. The point you select is the position of the landing head.

4 If you want to change the spacing of the multileader objects, enter **s** and specify one of the following spacing methods:
   - **Distribute**. Spaces content evenly between two selected points.
   - **Use Current**. Uses the current spacing between multileaders.
   - **Make Parallel**. Places content so that each of the last line segments in the selected multileaders are parallel.

5 In the drawing click a point to end the alignment.

Toolbar: Multileader
Command entry: MLEADERALIGN

To collect multiple notes to be attached to a single landing

1 Click Annotate tab ➤ Multileaders panel ➤ Collect. On the Multileader toolbar, click Collect Multileaders.

2 Select multileaders in the order you want them to be collected. The last multileader selected retains its landing. Press ENTER.

Toolbar: Multileader
Command entry: MLEADERCOLLECT

To create a landing line with multiple segments

1 Click Home tab ➤ Annotation panel ➤ Multileader Style.

2 In the Multileader Style Manager, click New to create a new multileader style.

3 In the Create New Multileader Style dialog box, specify a name for the new multileader style.

4 In the Modify Multileader Style dialog box, select the Leader Structure tab.

5 In the Constraints group box, select the Maximum Leader Points check box. In the box to the right, specify a maximum number of points to be prompted for upon creation of a new multileader. Click OK.

6 In the Multileader Style Manager, click Set Current to apply the new multileader style to new multileaders that you create.

Toolbar: Multileader

Command entry: MLEADERSTYLE

To change the properties of a multileader leader

1. Press CTRL and select a segment of the leader line.
2. Right-click and then select Properties from the shortcut menu.
3. In the Properties palette, specify the properties of the segment.

Toolbar: Standard 
Command entry: PROPERTIES

Quick Reference

Commands

DDEDIT

Edits single-line text, dimension text, attribute definitions, and feature control frames.

MLEADER

Creates a multileader object.

MLEADERALIGN

Aligns and spaces selected multileader objects.

MLEADERCOLLECT

Organizes selected multileaders that contain blocks into rows or columns, and displays the result with a single leader.

MLEADEREDIT

Adds leader lines to, or removes leader lines from, a multileader object.

MLEADERSTYLE

Creates and modifies multileader styles.

PROPERTIES

Controls properties of existing objects.

System Variables

DIMASSOC

Controls the associativity of dimension objects and whether dimensions are exploded.

DIMASZ

Controls the size of dimension line and leader line arrowheads.

DIMCLRD

Assigns colors to dimension lines, arrowheads, and dimension leader lines.

DIMGAP

Sets the distance around the dimension text when the dimension line breaks to accommodate dimension text.

DIMLDRBLK

Specifies the arrow type for leaders.

DIMSCALE

Sets the overall scale factor applied to dimensioning variables that specify sizes, distances, or offsets.

DIMTAD

Controls the vertical position of text in relation to the dimension line.

DIMTXTDIRECTION

Specifies the reading direction of the dimension text.

MLEADERSCALE

Sets the overall scale factor applied to multileader objects.

Utilities

No entries

Command Modifiers

No entries

Work with Leader Styles

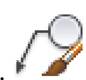
The appearance of a leader is controlled by its multileader style. You can use the default multileader style, STANDARD, or create your own multileader styles.

The multileader style can specify formatting for landing lines, leader lines, arrowheads, and content. For example, the STANDARD multileader style uses a straight leader line with a closed filled arrowhead and multiline text content.

NOTE Annotative blocks cannot be used as either content or arrowheads in multileader objects.

Once a multileader style has been defined, you can set it as the current multileader style to be used when the MLEADER command is invoked.

To define a leader style

1. Click Home tab ➤ Annotation panel ➤ Multileader Style. 
2. In the Multileader Style Manager, click New.
3. In the Create New Multileader Style dialog box, specify a name for the new multileader style.
4. In the Modify Multileader Style dialog box, Leader Format tab, select or clear the following options:
   - **Type**. Determines the type of landing. You can choose a straight landing, spline landing, or no landing.
   - **Color**. Determines the color of the landing.
   - **Linetype**. Determines the linetype of the landing.
   - **Lineweight**. Determines the lineweight of the landing.
5. Specify a symbol and size for the multileader arrowhead.

6 On the Leader Structure tab, select or clear the following options:

- **Maximum Leader Points**. Specifies a maximum number of points for the multileader landing line.
- **First and Second Segment Angles**. Specifies the angle of the first and second points in the landing.
- **Landing - Keep Horizontal**. Attaches a horizontal landing to the multileader content.
- **Set Landing Distance**. Determines the fixed distance for the multileader landing line.

7 On the Content tab, specify either text or block content for the multileader. If the multileader object will contain text content, then select or clear the following options:

- **Default Text**. Sets default text for the multileader content. A field can be inserted here.
- **Text Style**. Specifies a predefined text style for the attribute text. Currently loaded text styles are displayed.
- **Text Angle**. Specifies the rotation angle of the multileader text.
- **Text Color**. Specifies the color of the multileader text.
- **Paper Height**. Sets the height of the text as it will display in paper space.
- **Frame Text**. Frames the multileader text content with a text box.
- **Attachment**. Controls the attachment of the landing to the multileader text.
- **Landing Gap**. Specifies the distance between the landing and the multileader text.

If block content is specified, then select or clear the following options:

- **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, the insertion point, or the center point of the block.
- **Color**. Specifies the color of the multileader block content. ByBlock is selected by default.

8 Click OK.

Menu: Format ➤ Multileader Style

Toolbar: Multileader 
Command entry: MLEADERSTYLE

To apply a leader style to an existing leader

1. Select the multileader to which you want to apply a new style.
2. On the ribbon, click the Annotate tab, Multileaders panel. Select the desired multileader style from the drop-down list.
3. To create a new style, click the Multileader Style icon.

Toolbar: Multileader 
Command entry: MLEADERSTYLE

Quick Reference

Commands

MLEADERSTYLE

Creates and modifies multileader styles.

System Variables

CMLEADERSTYLE

Sets the name of the current multileader style.

Utilities

No entries

Command Modifiers

No entries

Add Content to a Leader

Leaders can contain multiline text or blocks to label parts of your drawing.

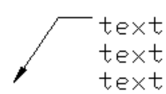
Leaders Containing Multiline Text

Leaders can contain multiline text as content. Text can be inserted by default when creating a leader style. Text style, color, height, and alignment can be applied and modified in leader annotations. You can also offset a multiline text object by specifying a landing gap distance in the current leader style.

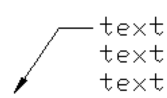
You can create annotative multileaders with text as content. The text content will be scaled according to the specified scale representation. Width, justification, attachment, and rotation settings for text content can be different depending on the specified scale representation. Actual text content cannot change with the scale representation.

There are several options for placing multiline text as content in a leader object.

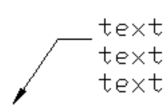
Top of top line



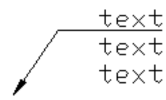
Middle of top line



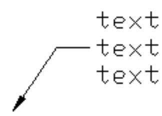
Bottom of top line



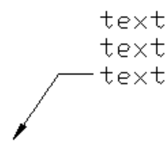
Underline top line



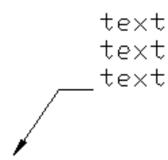
Middle of text



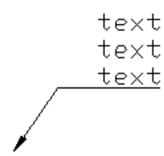
Middle of bottom line



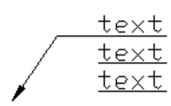
Bottom of bottom line



Underline bottom line



Underline all text



Leaders Containing Blocks

Multileaders can contain blocks as content by applying a multileader style that references a block in your drawing.

NOTE Annotative Blocks cannot be used as either content or arrowheads in multileader objects.

Blocks can be connected to a multileader by attaching the landing to a selected insertion point on the block. You can also connect a multileader to a center point on the selected block.

You can create annotative multileaders with blocks as content. The block content will be scaled according to the specified scale representation. Any attributes within the block content will not change with the scale representation. Non-annotative multileader objects can be scaled using the MLEADERSCALE system variable.

See also:

- Use Fields in Text on page 1532

Quick Reference

Commands

DDEDIT

Edits single-line text, dimension text, attribute definitions, and feature control frames.

DIMSTYLE

Creates and modifies dimension styles.

MLEADER

Creates a multileader object.

MTEXT

Creates a multiline text object.

OPTIONS

Customizes the program settings.

PROPERTIES

Controls properties of existing objects.

PURGE

Removes unused items, such as block definitions and layers, from the drawing.

QTEXT

Controls the display and plotting of text and attribute objects.

SPELL

Checks spelling in a drawing.

STYLE

Creates, modifies, or specifies text styles.

TEXT

Creates a single-line text object.

System Variables

DIMGAP

Sets the distance around the dimension text when the dimension line breaks to accommodate dimension text.

FONTALT

Specifies the alternate font to be used when the specified font file cannot be located.

FONTMAP

Specifies the font mapping file to be used.

MLEADERSCALE

Sets the overall scale factor applied to multileader objects.

MTEXTED

Sets the application for editing multiline text objects.

QTEXTMODE

Controls how text is displayed.

TEXTFILL

Controls the filling of TrueType fonts while plotting and rendering.

TEXTQLTY

Sets the resolution tessellation fineness of text outlines.

Utilities

No entries

Command Modifiers

No entries

Use Fields in Text

A field is updatable text that is set up to display data that may change during the life cycle of the drawing. When the field is updated, the latest value of the field is displayed.

Insert Fields

A field is text that contains instructions to display data that you expect to change during the life cycle of the drawing.

When a field is updated, the latest data is displayed. For example, the value of the FileName field is the name of the file. If the file name changes, the new file name is displayed when the field is updated.

Fields can be inserted in any kind of text (except tolerances), including text in table cells, attributes, and attribute definitions. When any text command is active, Insert Field is available on the shortcut menu.

Some sheet set fields can be inserted as placeholders. For example, you can insert SheetNumberAndTitle as a placeholder. Later, when the layout is added to a sheet set, the placeholder field displays the correct sheet number and title.

Block placeholder fields can be used in block attribute definitions while you're working in the Block Editor.

A field for which no value is available displays hyphens (----). For example, the Author field, which is set in the Drawing Properties dialog box, may be blank. An invalid field displays pound signs (####). For example, the CurrentSheetName field, which is valid only in paper space, displays pound signs if it is placed in model space.

Change the Appearance of a Field

The field text uses the same text style as the text object in which it is inserted. By default, fields are displayed with a light gray background that is not plotted (FIELDDISPLAY system variable).

Formatting options in the Field dialog box control the appearance of the text that is displayed. The options that are available depend on the type of field. For example, the format for date fields includes options for displaying the day of the week and the time, and the format for named object fields includes capitalization options.

Edit a Field

A field is part of a text object and it can be edited from a text editor. The easiest way to edit a field is to double click the text object that contains the field and then, to display the Field dialog box, double click the field. These operations are available on the shortcut menus as well.

If you no longer want to update a field, you can preserve the value that is currently displayed by converting the field to text.

The field expression, consisting of escape characters and a field code, is shown in the Field dialog box but cannot be edited.

To insert a field in text

1. Double-click the text to display the appropriate text editing dialog box.
2. Place the cursor where you want the field text to appear and right-click. Click Insert Field.

   For keyboard access, press CTRL+F.
3. In the Field dialog box, in Field Category, select All or select a category.

   The fields in the selected category are displayed in the Field Names list.
4. In the Field Names list, select a field.

   The current value of most fields is displayed in a shaded text box to the right of Field Category. The current value of a date field is displayed in the Examples list.
5. Select a format and any other options.

   For example, when the NamedObject field is selected, you select a type (for example, layer or textstyle) and a name (for example, 0 for layer or STANDARD for textstyle).

   Field Expression displays the expression that underlies the field. The field expression cannot be edited, but you can learn how fields are constructed by viewing this area.
6. Click OK to insert the field.

The field displays its current value in the text when the Field dialog box closes.

To insert a field in a table

1. Double-click inside a cell in a table to select it for editing.
2. Place the cursor where you want the field text to appear and right-click. Click Insert Field.
3. In the Field dialog box, select All or select a category.
4. In the Field Names list, select a field.

   The current value of the field is displayed in a shaded text box to the right of Field Category.
5. Select a format and any other option.
6. Click OK to insert the field.

   The field displays its current value when you move to the next cell.

To use a field to display a property of an object

1. Double-click a text object to display the appropriate text editing dialog box.
2. Place the cursor where you want the field text to appear and right-click. Click Insert Field.
3. In the Field dialog box, in Field Category, select All.
4. In the Field Names list, select Object.
5. In Object Type, click the Select Object button, and select an object in the drawing.
6. In the Field dialog box, in Property, select the property whose value you want to display in the text.

   For example, the field could display the radius of a selected circle.
7. Select a format for the text.
8. Click OK.

   The current value for the object's property is displayed in the text.

To insert a sheet set placeholder field

1 Click Home tab ➤ Block panel ➤ Define Attributes. 

2 In the Attribute Definition dialog box, under Mode, click Preset. Specify any text options.

3 Under Attribute, in the Tag box, enter a name for the field.

4 To the right of the Value box, click the Insert Field button.

5 In the Field dialog box, under Field Category, select SheetSet.

6 In the Field Names list, select SheetSetPlaceholder.

7 In Placeholder Type, select the type of field that you want.

   Temporary Value previews the placeholder field.

8 In Format, select a capitalization style for the placeholder field.

   When the field is updated, the same style is used for the value of the field.

9 (Optional) Click Associate Hyperlink if you want the field to serve as a hyperlink to a location.

   NOTE Associate Hyperlink is not available for the ViewportScale field.

   ---

10 Click OK to exit the Field dialog box.

   The placeholder field displays its name when the Field dialog box closes: for example, SheetNumber. Later, when the block is inserted from the Views List tab shortcut menu in the Sheet Set Manager, the field displays a value based on the sheet it is dragged to: for example, its sheet number.

11 Click OK in the Attribute Definition dialog box. Specify the location for the field text to appear near the geometry that you have created to use with the Sheet Set Manager as a callout block or a label block.

12 Create a block that includes the newly created attribute object along with the geometry for the block.

Command entry: ATTDEF

To format a field value

1. Double-click a text object to display the appropriate text editing dialog box.
2. Double-click the field you want to format.

   The Field dialog box is displayed. If formatting is available for the field, the Field Format button is displayed.
3. Click Field Format.

   In the Additional Format dialog box, the current value of the field is displayed. When you select an option, the result is displayed in Preview.
4. Enter a conversion factor to apply to the current value.

   For example, to convert inches to millimeters, enter **0.03937**.
5. Enter any text that you want to precede or follow the field value.

   For example, enter **mm** for millimeters.
6. Select a decimal separator. Select None or Comma to group thousands.
7. Select an option for suppressing zeros:
   - *Leading*: Suppresses leading zeros in all decimal field values. For example, 0.5000 becomes .5000.
   - *Trailing*: Suppresses trailing zeros in all decimal field values. For example, 12.5000 becomes 12.5, and 30.0000 becomes 30.
   - *0 Feet*: Suppresses the feet portion of a feet-and-inches field 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 field value when the distance is an integral number of feet. For example, 1'-0" becomes 1'.
8. Click OK.

   In the Field dialog box, the field value is displayed in Preview with the formatting you specified.
9. Click OK.

To edit a field

1. Double-click a text object to display the appropriate text editing dialog box.

2 Double-click the field that you want to edit.

   The Field dialog box is displayed.

3 Make any needed changes.

4 Click OK to exit the Field dialog box.

5 Exit the text editor.

Quick Reference

Commands

FIELD

Creates a multiline text object with a field that can be updated automatically as the field value changes.

FIND

Finds the text that you specify, and can optionally replace it with other text.

INSERT

Inserts a block or drawing into the current drawing.

LIST

Displays property data for selected objects.

MTEXT

Creates a multiline text object.

SPELL

Checks spelling in a drawing.

TABLE

Creates an empty table object.

TABLEEXPORT

Exports data from a table object in CSV file format.

TABLESTYLE

Creates, modifies, or specifies table styles.

UPDATEFIELD

Manually updates fields in selected objects in the drawing.

System Variables

CTABLESTYLE

Sets the name of the current table style.

FIELDDISPLAY

Controls whether fields are displayed with a gray background.

FIELDEVAL

Controls how fields are updated.

Utilities

No entries

Command Modifiers

No entries

Update Fields

When a field is updated, it displays the latest value. You can update fields individually or update all fields in one or more selected text objects.

You can also set fields to be updated automatically when the drawing is opened, saved, plotted, regenerated, or sent through ETRANSMIT.

Settings on the User Preferences Tab (Options Dialog Box) control whether fields are updated automatically or on demand (FIELDEVAL system variable). The Date field cannot be updated automatically regardless of the setting of FIELDEVAL.

NOTE When the DEMANDLOAD system variable is set to 2, fields cannot be updated until you use FIELD or UPDATEFIELD.

Contextual Fields in Blocks and Xrefs

Some fields are contextual; that is, their value is different depending on which space or layout they reside in. For example, because each layout can have a different page setup attached, the value displayed by the PlotOrientation field can be different in different layouts in the same drawing.

List of contextual fields	
CurrentSheetCustom	CurrentSheetTitle
CurrentSheetDescription	DeviceName
CurrentSheetNumber	PaperSize
CurrentSheetNumberAndTitle	PlotDate
CurrentSheetSet	PlotOrientation
CurrentSheetSetCustom	PlotScale
CurrentSheetSetDescription	PageSetupName
CurrentSheetSubSet	PlotStyleTable

For compatibility with previous versions, contextual fields in blocks and xrefs are not updated when you insert them in a drawing; instead, the field displays the last cached value. Therefore, if you want to use a contextual field within a block, for example, a title block, you must insert the field as an attribute. For example, a title block can use the CurrentSheetNumber field as an attribute. When you insert the title block, the field displays the sheet number of the sheet on which the title block is inserted.

Most fields are not contextual and are updated in blocks and xrefs. Fields in xrefs are updated based on the host file, not the source xref. These fields do not have to be placed in attributes. For example, a field that displays the sheet number of a particular sheet in a sheet set and that updates if that sheet number changes, is a property of the sheet set. When you create the field, you select the SheetSet field name, select the sheet set and the sheet that you want in the tree view, and then select the property SheetNumber for the field value to be displayed. This field displays the sheet number of that sheet, even if you put the field in a block and insert it in another drawing. If the sheet is removed from the sheet set, it no longer has a sheet number, and the field becomes invalid and displays pound signs.

Some sheet set fields can be inserted as placeholders. For example, when you create your own callout blocks and label blocks, you can insert the SheetNumber field as a placeholder. Later, when the block is inserted from the Views List tab shortcut menu in the Sheet Set Manager, the field displays the sheet number of the drawing.

Compatibility with Previous Releases

When a drawing with fields is opened in AutoCAD 2004 or earlier, the fields are not updated; they display the value last displayed in the drawing before it was opened. If no changes are made to a field, it is updated normally when it is reopened in a release that supports fields.

Fields are not available in the previous releases of AutoCAD LT. When a drawing with fields is opened in the previous releases of AutoCAD LT, the fields are evaluated based on the setting of the FIELDEVAL system variable in the drawing, but the FIELDEVAL system variable is not accessible.

See also:

- Work with AutoCAD Drawings in AutoCAD LT

To update a field manually

1. Double-click text.
2. Select the field to update and right-click. Click Update Field.

To update multiple fields manually

1. Click Blocks & References tab ➤ Data panel ➤ Update Fields .
2. At the Select Objects prompt, select the objects that contain the fields you want to update and press ENTER.

   All of the fields in the selected objects are updated.

Command entry: UPDATEFIELD

To update fields automatically

1. At the Command prompt, enter **fieldeval**.
2. Enter a bitcode that is the sum of any 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

For example, to update fields only when the file is opened, saved, or plotted, enter **7**.

Quick Reference

Commands

FIELD

Creates a multiline text object with a field that can be updated automatically as the field value changes.

UPDATEFIELD

Manually updates fields in selected objects in the drawing.

System Variables

FIELDDISPLAY

Controls whether fields are displayed with a gray background.

FIELDEVAL

Controls how fields are updated.

Utilities

No entries

Command Modifiers

No entries

Use Hyperlinks in Fields

The Hyperlink field assigns a hyperlink to any piece of text.

The hyperlink works the same way as a hyperlink attached to an object. When the cursor pauses over the text, a hyperlink cursor is displayed, along with a tooltip that describes the hyperlink. Hold down the CTRL key and click to follow the link.

NOTE The Hyperlink field uses an absolute path to a file; the HYPERLINK command can create a hyperlink with a relative path.

The fields that point to sheet and view titles and numbers can be assigned a hyperlink when they are created. When these items are changed or moved in the Sheet Set Manager, the hyperlinks associated with them still jump to the correct location.

Hyperlinks in fields are converted from links to DWG files into links to DWF files when published to a multi-sheet DWF file format.

To add a hyperlink field to text

1 Click Home tab ➤ Annotation panel ➤ Multiline Text. 

2 Place the cursor where you want the hyperlink text to appear.

3 Right-click in the editor. Click Insert Field.

4 In the Field dialog box, in Field Category, select Linked.

5 In Field Names, select Hyperlink, and click Hyperlink.

6 In the Insert Hyperlink dialog box, use one of the following methods to specify a location:
   - Under Type the File or Web Page Name, enter the path and name of the file that you want to associate with the hyperlink.
   - Under Browse For, click File, Web Page, or Target. Navigate to the location to which you want to link. Click Open or OK.

7 (Optional) In Text to Display, select the default text that is displayed, and enter the link text that you want to appear in the mtext object.

8 Click OK to close each dialog box.

9 To save your changes and exit the editor, use one of the following methods:
   - Click OK on the toolbar.
   - Click in the drawing outside the editor.

- Press CTRL+ENTER.

The hyperlink is displayed in the mtext object with the link text that you entered. Use CTRL+click to jump to the hyperlinked location.

Toolbar: Draw
Command entry: MTEXT

Quick Reference

Commands

FIELD

Creates a multiline text object with a field that can be updated automatically as the field value changes.

FIND

Finds the text that you specify, and can optionally replace it with other text.

INSERT

Inserts a block or drawing into the current drawing.

LIST

Displays property data for selected objects.

MTEXT

Creates a multiline text object.

SPELL

Checks spelling in a drawing.

TABLE

Creates an empty table object.

TABLEEXPORT

Exports data from a table object in CSV file format.

TABLESTYLE

Creates, modifies, or specifies table styles.

UPDATEFIELD

Manually updates fields in selected objects in the drawing.

System Variables

CTABLESTYLE

Sets the name of the current table style.

FIELDDISPLAY

Controls whether fields are displayed with a gray background.

FIELDEVAL

Controls how fields are updated.

Utilities

No entries

Command Modifiers

No entries

Work with Text Styles

When you enter text into your drawing, the current text style determines the text font, size, angle, orientation, and other text characteristics.

Overview of Text Styles

All text in a drawing has a text style associated with it. When you enter text, the program uses the current text style.

The current text style sets the font, size, obliquing angle, orientation, and other text characteristics. If you want to create text using a different text style, you can make another text style current. The table shows the settings for the STANDARD text style.

The settings for the current text style are displayed at the Command prompts. You can use or modify the current text style or create and load a new text style. Once you've created a text style, you can modify its characteristics, change its name, or delete it when you no longer need it.

Create and Modify Text Styles

Except for the default STANDARD text style, you must create any text style that you want to use.

Text style names can be up to 255 characters long. They can contain letters, numbers, and the special characters dollar sign ($), underscore (_), and hyphen (-). If you don't enter a text style name, the text styles are automatically named Stylen, where *n* is a number that starts at 1.

You can modify an existing text style in the Text Style dialog box by changing the settings. You can also update existing text of that text style to reflect the changes.

Certain style settings affect multiline and single-line text objects differently. For example, changing the Upside Down and Backwards options has no effect on multiline text objects. Changing Width Factor and Obliquing has no effect on single-line text.

If you rename an existing text style, any text using the old name assumes the new text style name.

You can remove unreferenced text styles from your drawing with PURGE or by deleting the text styles from the Text Styles dialog box. The STANDARD text style cannot be removed.

Change Text Style

When you change the text style of a multiline text object, the updated settings are applied to the entire object, and some formatting of individual characters might not be retained. The following table describes the effects of text style change on character formatting.

Formatting	Retained?
Bold	No
Color	Yes
Font	No
Height	No
Italic	No
Stacking	Yes

Formatting	Retained?
Underlining	Yes

Annotative Text Styles

Use text for notes and labels in your drawing. You create annotative text by using an annotative text style, which sets the height of the text on the paper.

For more information about creating and working with an annotative text, see Create Annotative Text on page 1404.

See also:

- Scale Annotations on page 1393

To set the current text style

- On the ribbon, click the Annotate tab ➤ Text panel. in the Text Style drop-down list, select a text style.

Command entry: STYLE

Quick Reference

Commands

PURGE

Removes unused items, such as block definitions and layers, from the drawing.

STYLE

Creates, modifies, or specifies text styles.

System Variables

FONTALT

Specifies the alternate font to be used when the specified font file cannot be located.

FONTMAP

Specifies the font mapping file to be used.

TEXTSIZE

Sets the default height for new text objects drawn with the current text style.

TEXTSTYLE

Sets the name of the current text style.

Utilities

No entries

Command Modifiers

No entries

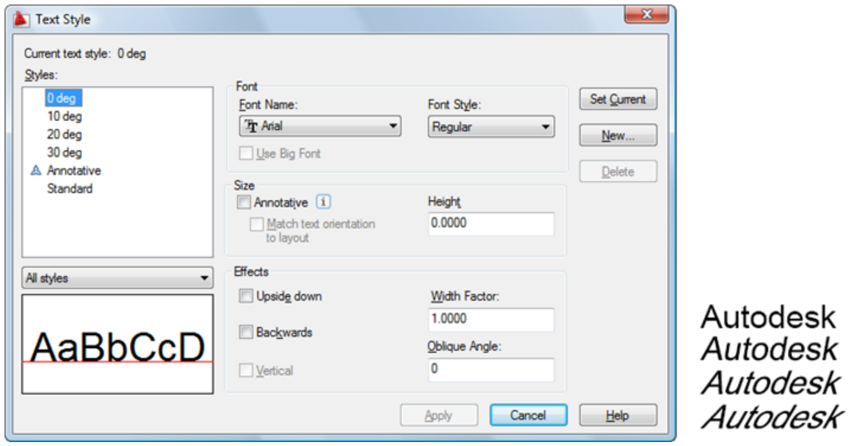
Assign Text Fonts

You can assign a text font as part of the text style definition. Several factors depend on the type of text you are working with.

Overview of Assigning Text Fonts

Fonts define the shapes of the text characters that make up each character set. You can use TrueType fonts in addition to compiled SHX fonts.

A single font can be used by more than one text style. If your company has a standard font, you can modify other text style settings to create a set of text styles that use this standard font in different ways. The following illustration shows the same font used by different text styles that use different obliquing settings to define the slant of the text.



You can assign a font to a text style by selecting a font file from the list in the Text Style dialog box.

Quick Reference

Commands

STYLE

Creates, modifies, or specifies text styles.

System Variables

FONTALT

Specifies the alternate font to be used when the specified font file cannot be located.

FONTMAP

Specifies the font mapping file to be used.

Utilities

No entries

Command Modifiers

No entries

Use TrueType Fonts

Several factors affect the display of TrueType fonts in a drawing.

TrueType fonts always appear filled in your drawing; however, when you plot, the TEXTFILL system variable controls whether the fonts are filled. By default TEXTFILL is set to 1 to plot the filled-in fonts.

The In-Place Text Editor can display only fonts that are recognized by Microsoft Windows. Because SHX fonts are not recognized by Windows, a TrueType equivalent is supplied in the In-Place Text Editor when you select an SHX or any other non-TrueType font for editing.

See also:

- Set Text Height on page 1556

To assign a TrueType font to a text style

1. Click Annotate tab ➤ Text panel ➤ Panel Launcher button.
2. In the Text Style dialog box under Style Name, click New.
3. In the New Text Style dialog box, enter a style name for the new text style. Click OK.
4. Under Font Name, select a TrueType font from the list.

   TrueType fonts display a TrueType icon in front of their names.
5. To update text of the current style in the drawing, click Apply.
6. Click Close.

Menu: Format ➤ Text Style

Toolbar: Styles 

Command entry: STYLE

Quick Reference

Commands

QTEXT

Controls the display and plotting of text and attribute objects.

STYLE

Creates, modifies, or specifies text styles.

System Variables

QTEXTMODE

Controls how text is displayed.

TEXTFILL

Controls the filling of TrueType fonts while plotting and rendering.

TEXTQLTY

Sets the resolution tessellation fineness of text outlines.

Utilities

No entries

Command Modifiers

No entries

Use Text Fonts for International Work

Several factors affect your choosing, entering, and displaying international text in a drawing.

The program supports the Unicode character-encoding standard. An SHX font encoded using the Unicode standard font can contain many more characters than are defined in your system; therefore, to use a character not directly available from the keyboard, you can enter the escape sequence \U+*nnnn*, where *nnnn* represents the Unicode hexadecimal value for the character.

Beginning with AutoCAD 2007, all SHX shape fonts are encoded with the Unicode standard with the exception of Big Fonts. When choosing a text font for international work, you can use either a TrueType Font or a Big Font.

Asian Big Font SHX Files

Asian alphabets contain thousands of non-ASCII characters. To support such text, the program provides a special type of shape definition known as a Big Font file. You can set a style to use both regular and Big Font files.

Asian Language Big Fonts Included in the Product

Font File Name	Description
@extfont2.shx	Japanese vertical font (a few characters are rotated to work correctly in vertical text)
bigfont.shx	Japanese font, subset of characters
chineset.shx	Traditional Chinese font
extfont.shx	Japanese extended font, level 1
extfont2.shx	Japanese extended font, level 2
gbcbig.shx	Simplified Chinese font
whgdtxt.shx	Korean font
whgtxt.shx	Korean font
whtgtxt.shx	Korean font
whtmtxt.shx	Korean font

When you specify fonts using -STYLE, the assumption is that the first name is the normal font and the second (separated by a comma) is the Big Font. If you enter only one name, it's assumed that it is the normal font and any associated Big Font is removed. By using leading or trailing commas when specifying the font file names, you can change one font without affecting the other, as shown in the following table.

Specifying fonts and Big Fonts at the Command prompt

Enter this ...	To specify this ...
[font name],[big font name]	Both normal fonts and Big Fonts
[font name],	Only a normal font (Big Font unchanged)

Specifying fonts and Big Fonts at the Command prompt	
Enter this ...	**To specify this ...**
,[big font name]	Only a Big Font (normal font unchanged)
[font name]	Only a normal font (Big Font, if any, removed)
ENTER (null response)	No change

NOTE Long file names that contain commas as font file names are not accepted. The comma is interpreted as a separator for an SHX font-Big Font pair.

See also:

- Substitute Fonts on page 1553

To assign an Asian-language SHX font to a text style

1. Click Home tab ➤ Annotation panel ➤ Text Style.
2. In the Text Style dialog box, under Style Name, click New.
3. In the New Text Style dialog box, enter a style name for the new text style. Click OK.
4. Under Font Name, select the name of an SHX font file, and then select Use Big Font to select an Asian-language big font.

   When you select Use Big Font, the Font Style box changes to a Big Font Name box. Only SHX fonts are available for selection, and only Big Font names are shown in the Big Font box.
5. To see the effects on different characters, enter a text string in the sample text box that is located to the left of the Preview button. Click Preview.
6. To update text of the current style in the drawing, click Apply.
7. Click Close.

Toolbar: Styles
Command entry: STYLE

Quick Reference

Commands

STYLE

Creates, modifies, or specifies text styles.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Substitute Fonts

A font used in a drawing but that is not currently available on your system is automatically substituted with another font.

The program accommodates a font that is not currently on your system by substituting another font.

Specify an Alternate Font

If your drawing specifies a font that is not currently on your system, the font designated as your alternate font is automatically substituted. By default, the *simplex.shx* file is used. If you want to specify a different font, enter the alternate font file name by changing the FONTALT system variable. If you use a text style that uses a Big Font, you can map it to another font using the FONTALT system variable. This system variable uses a default font file pair: *txt.shx* and *bigfont.shx*. For more information, see Use Text Fonts for International Work on page 1550.

In previous releases, you could display PostScript® fonts in the drawing. Because later releases cannot display PostScript fonts, Autodesk has supplied TrueType font equivalents. These PostScript fonts are mapped to the equivalent TrueType fonts in a font mapping file. Additionally, when a TrueType font is not available, you can specify a different TrueType font, making sure that the fonts are similar to avoid text length or wrapping problems.

If the default font does not support the characters you enter using the In-Place Text Editor (MTEXT command), an alternative font is substituted.

CIF or MIF codes entered with the In-Place Text Editor or with the DTEXT command are now automatically converted to display the actual characters.

Edit the Font Mapping File

A font mapping file is a list of text fonts and their substitutes. If a text font used in a drawing cannot be located, another text font is substituted for the missing font using a font mapping file.

Each line in the font mapping file contains the name of a font file (with no file extension or path) followed by a semicolon (;) and the name of the substitute font file. The substitute file name includes a file extension such as *.ttf*.

A font mapping file is an ordinary ASCII text file with a *.fmp* extension. The default font mapping file is *acad.fmp* for AutoCAD, and *acadlt.fmp* for AutoCAD LT. You can change the font assignments in a font mapping file using any ASCII text editor.

For example, you could use the following entry in a font map file to specify that the *timesnr.pfb* font file be substituted with the *times.ttf* font file:

timesnr;times.ttf
The following table shows the font substitution rules used if a font file cannot be located when a drawing is opened.

Font substitution

File extension	First mapping order	Second mapping order	Third mapping order	Fourth mapping order
.ttf	Use font mapping table	Use font defined in text style	Windows substitutes a similar font	
.shx	Use font mapping table	Use font defined in text style	Use FONTALT	Prompt for new font
.pfb	Use font mapping table	Use FONTALT	Prompt for new font	

Display Proxy Fonts

For third-party or custom SHX fonts that have no TrueType equivalent, one of several different TrueType fonts called proxy fonts is substituted. In the

In-Place Text Editor, proxy fonts look different from the fonts they represent to indicate that the proxy fonts are substitutions for the fonts used in the drawing.

Custom SHX fonts do not appear in the Font list on either the MTEXT Ribbon Contextual Tab or the Text Formatting toolbar. If you want to format characters by assigning one of these fonts, first create a text style that uses the font and then apply that text style to the characters.

To specify a font mapping file

1 Click Tools menu ➤ Options.

2 On the Files tab, in the list, double-click Text Editor, Dictionary, and Font File Names.

3 Double-click Font Mapping File.

   The *acad.fmp* file is specified by default.

4 To change the font mapping file, double-click the arrow line to open the Select a File dialog box. Select a file and click Open. Click OK.

5 At the Command prompt, enter **regen** to convert existing text using the new font mappings.

Command entry: OPTIONS

To specify a default alternate font

1 At the Command prompt, enter **fontalt**.

2 Enter the name of the font file you want to use as the alternative.

Quick Reference

Commands

MTEXT

Creates a multiline text object.

OPTIONS

Customizes the program settings.

System Variables

FONTALT

Specifies the alternate font to be used when the specified font file cannot be located.

FONTMAP

Specifies the font mapping file to be used.

Utilities

No entries

Command Modifiers

No entries

Set Text Height

Text height determines the size in drawing units of the letters in the font you are using.

The exception is TrueType fonts: the value usually represents the size of the uppercase letters.

If you specify a fixed height as part of a text style, the Height prompt is bypassed when you create single-line text. When the height is set to 0 in the text style, you are prompted for the height each time you create single-line text. Set the value to 0 if you want to specify the height as you create text.

TrueType Fonts

For TrueType fonts, the value specified for text height represents the height of a capital letter plus an ascent area reserved for accent marks and other marks used in non-English languages. The relative portion of text height that is assigned to capital letters and ascent characters is determined by the font designer at the time the font is designed; consequently, it varies from font to font.

In addition to the height of a capital letter and the ascent area that make up the text height specified by the user, TrueType fonts have a descent area for portions of characters that extend below the text insertion line, for example, *y*, *j*, *p*, *g*, and *q*.

When you apply a text height override to all text in the editor, the entire multiline text object is scaled, including its width.

To set text height in a text style

1 Click Home tab ➤ Annotation panel ➤ Text Style.

2 In the Text Style dialog box, select a style from the Style Name list.

3 Under Font, enter the text height (in drawing units) in the Height box.

4 To update existing text that uses this text style, click Apply.

5 Click Close.

Toolbar: Styles
Command entry: STYLE

Quick Reference

Commands

STYLE

Creates, modifies, or specifies text styles.

System Variables

TEXTSIZE

Sets the default height for new text objects drawn with the current text style.

TEXTSTYLE

Sets the name of the current text style.

Utilities

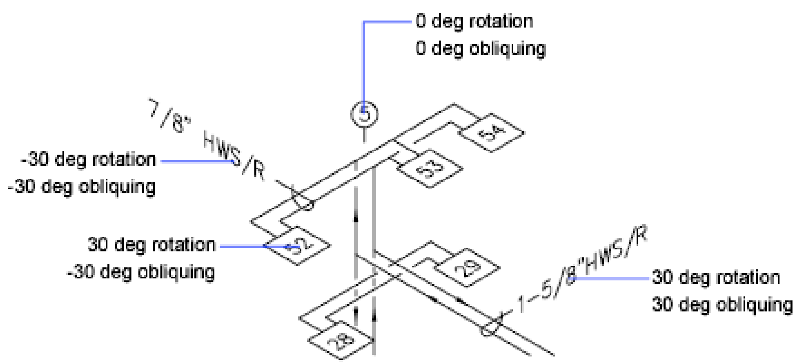
No entries

Command Modifiers

No entries

Set Text Obliquing Angle

The obliquing 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.

To set the obliquing angle in a text style

1 Click Home tab ➤ Annotation panel ➤ Text Style. 

2 In the Text Style dialog box, select a text style from the Style Name list.

3 Under Effects, enter an angle between -85 and 85 in the Oblique Angle box.

   A positive value slants text to the right. A negative value slants text to the left.

4 To update existing text that uses this text style, click Apply.

5 Click Close.

Toolbar: Styles
Command entry: STYLE

Quick Reference

Commands

STYLE

Creates, modifies, or specifies text styles.

System Variables

No entries

Utilities

No entries

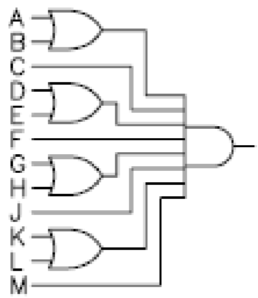
Command Modifiers

No entries

Set Horizontal or Vertical Text Orientation

Text can be vertical or horizontal. Text can have a vertical orientation only if the associated font supports dual orientation.

Lines of text are oriented to be vertical or horizontal. Text can have a vertical orientation only if the associated font supports dual orientation. You can create more than one line of vertical text. Each successive text line is drawn to the right of the preceding line. The normal rotation angle for vertical text is 270 degrees.



vertical text

NOTE Vertical orientation is not supported for TrueType fonts and symbols.

Vertical Text for Asian Languages

- **SHX fonts.** Text can be created with SHX fonts and Big Fonts for vertical display in the same way as for previous releases. For best results, use the single-line TEXT command, not MTEXT. You can select a vertical style in the Text Style dialog box.
- **TrueType fonts.** You still select fonts starting with the @ sign, but now the text is automatically rotated 270 degrees. (In AutoCAD 2005 and earlier versions, you had to manually rotate this text.) Vertical cursor movement is now supported for vertical text.

To set vertical orientation in a text style

1. Click Home tab ➤ Annotation panel ➤ Text Style. 
2. In the Text Style dialog box, select a text style from the Style Name list.
3. Under Effects, select Vertical.
4. To update existing text that uses this text style, click Apply.
5. Click Close.

Toolbar: Styles 
Command entry: STYLE

Quick Reference

Commands

STYLE

Creates, modifies, or specifies text styles.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Change Text

You can change text content, formatting, and properties such as scale and justification.

Overview of Changing Text

Text, whether created with TEXT, MTEXT, or MLEADER can be modified like any other object.

You can move, rotate, erase, and copy it. You can change text properties in the Properties palette.

You can also edit the contents of existing text and create a mirror image of it. The MIRRTEXT system variable controls whether text is also reversed when you mirror objects in your drawing. The procedures for modifying text vary slightly, depending on how the text was created.

Quick Reference

Commands

DDEDIT

Edits single-line text, dimension text, attribute definitions, and feature control frames.

FIND

Finds the text that you specify, and can optionally replace it with other text.

PROPERTIES

Controls properties of existing objects.

System Variables

MIRRTEXT

Controls how the MIRROR command reflects text.

TEXTFILL

Controls the filling of TrueType fonts while plotting and rendering.

TEXTQLTY

Sets the resolution tessellation fineness of text outlines.

Utilities

No entries

Command Modifiers

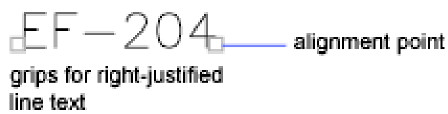
No entries

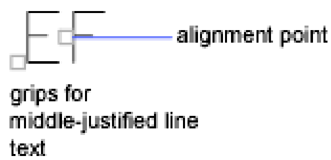
Change Single-Line Text

You can change the contents, formatting and properties of single-line text.

You can change single-line text with DDEDIT and PROPERTIES. Use DDEDIT when you need to change only the content of the text, not the formatting or properties of the text object. Use PROPERTIES when you want to change content, text style, location, orientation, size, justification, and other properties.

Text objects also have grips for moving, scaling, and rotating. A text object has grips at the lower-left corner of the baseline and at the alignment point.





The effect of a command depends on which grip you choose.

To edit single-line text

1. Click Modify menu ➤ Object ➤ Text ➤ Edit.
2. Select a single-line text object.
3. In the in-place editor, enter the new text.
4. Press ENTER.
5. Select another text object to edit, or press ENTER to end the command.

Command entry: DDEDIT

To modify properties of single-line text objects

1. Select a single-line text object.
2. Right-click the selected object. Click Properties.
3. In the Properties palette, enter any new text, and then change formatting and other properties as needed.

Toolbar: Standard 
Command entry: PROPERTIES

Quick Reference

Commands

DDEDIT

Edits single-line text, dimension text, attribute definitions, and feature control frames.

PROPERTIES

Controls properties of existing objects.

TEXT

Creates a single-line text object.

System Variables

DTEXTED

Specifies the user interface displayed for editing single-line text.

TEXTFILL

Controls the filling of TrueType fonts while plotting and rendering.

TEXTQLTY

Sets the resolution tessellation fineness of text outlines.

Utilities

No entries

Command Modifiers

No entries

Change Multiline Text

You can change the location and content of multiline text objects with the Properties palette, the In-Place Text Editor, and grips.

After you create multiline text, you can use the Properties palette to change

- Text style assignment
- Justification

- Width
- Rotation
- Line spacing

In addition, you can use either the MTEXT ribbon contextual tab (if the ribbon is active) or the In-Place Text Editor (if the ribbon is not active) to modify individual formatting, such as boldface and underlining, and to change the width of the multiline text object.

Change Text Location

You can use grips to move multiline text or to resize the line width. A multiline text object has grips at the four corners of the text boundary and, in some cases, at the justification point.

If you use the Properties palette to move multiline text, you can edit content and change properties at the same time.

Commands such as DIMLINEAR or LEADER create multiline text automatically without requiring that a bounding box be specified; these objects have only a single grip at the justification point.

When you need to align or move multiline text objects, you can use the Node and Insertion object snaps for precision. If the OSNAPNODELEGACY system variable is set to 0, the Node object snap ignores multiline text.

See also:

- Work with Text Styles on page 1544
- Control the Display of Polylines, Hatches, Gradient Fills, Lineweights, and Text on page 623

To change multiline text

1. Select a multiline text object.
2. Right-click the selected object. Click Properties.
3. In the Properties palette, enter any new text and change formatting and other settings as needed.

Toolbar: Standard 

Command entry: PROPERTIES

To change the width of a multiline text object

1. Double-click the multiline text object.
2. In the In-Place Text Editor, use one of the following methods:
   - Move the cursor over the right end of the ruler until the cursor changes to a double arrow. As you drag to the right to stretch the ruler, a tooltip displays the width. Release to set a new width.
   - Right-click the bottom of the ruler. Click Set Mtext Width. In the dialog box, enter the width in drawing units.
3. To save your changes and exit the editor, use one of the following methods:
   - On the MTEXT ribbon contextual tab, in the Close panel, click Close Text Editor.
   - Click in the drawing outside the editor.
   - Press CTRL+ENTER.

Quick Reference

Commands

DDEDIT

Edits single-line text, dimension text, attribute definitions, and feature control frames.

FIND

Finds the text that you specify, and can optionally replace it with other text.

MTEDIT

Edits multiline text.

PROPERTIES

Controls properties of existing objects.

STYLE

Creates, modifies, or specifies text styles.

System Variables

CENTERMT

Controls how grips stretch multiline text that is centered horizontally.

MIRRTEXT

Controls how the MIRROR command reflects text.

MTEXTED

Sets the application for editing multiline text objects.

OSNAPNODELEGACY

Controls whether the Node object snap can be used to snap to multiline text objects.

TEXTFILL

Controls the filling of TrueType fonts while plotting and rendering.

TEXTQLTY

Sets the resolution tessellation fineness of text outlines.

Utilities

No entries

Command Modifiers

No entries

Find and Replace Text

You can easily find and replace text with the FIND command

To search for and replace text, use FIND. Replacement is based on text content only; character formatting and text properties are not changed.

When searching for text in a 3D environment, the viewport will temporarily change to a 2D viewport so that text isn't blocked by 3D objects in your drawing.

With FIND, you can use wild-card characters in your search.

Character	Definition
# (Pound)	Matches any numeric digit

Character	Definition
@ (At)	Matches any alphabetic character
. (Period)	Matches any nonalphanumeric character
* (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
~ (Tilde)	Matches anything but the pattern; for example; ~*AB*matches all strings that don't contain AB
[ ]	Matches any one of the characters enclosed; for example, [AB]C matches AC and BC
[~]	Matches any character not enclosed; for example, [~AB]C matches XC but not AC
[-]	Specifies a range for a single character; for example, [A-G]C matches AC, BC, and so on to GC, but not HC
` (Reverse quote)	Reads the next character literally; for example, `~AB matches ~AB

To search for drawing files that contain a specific word or phrase, use the Search tool in Microsoft® Windows® . You can use the Search tool to find all textual data except text in tables and fields, and xrefs within drawing files.

To find specified text in a drawing

1. Click Annotate tab ➤ Text panel ➤ Find Text. 

2. In Find What, enter the text you want to find.
3. In Find Where, specify the parts of the drawing to search, or click the Select Objects button to select one or more text objects.
4. Click the Expand Find Options button to specify search options and text types for the specified text.

5 Click Find.

6 Use one of the following options to view the results of your search:

- To list all results in a table, click the List Results check box.
- To zoom to and highlight each result individually, leave the List Results check box unchecked.

7 Click Close.

Toolbar: Text Formatting
Command entry: FIND
Menu: Click Edit ➤ Find.

To replace text using the Find and Replace Dialog box

1 Click Annotate tab ➤ Text panel ➤ Find Text.

2 In Find What, enter the text you want to find.

3 In Find Where, specify the parts of the drawing to search, or click the Select Objects button to select one or more text objects.

4 Click the Expand Find Options button to specify search options and text types for the specified text.

5 In Replace With, enter the text with which you want to replace the found text.

6 Click Find.

7 Use one of the following options to view the results of your search:

- To list all results in a table, click the List Results check box.
- To zoom to and highlight each result individually, leave the List Results check box unchecked.

8 Use one of the following methods to replace text:

- To replace only the found instance of the text string, click Replace.
- To replace all instances of the text in Find Text String, click Replace All.

- If search results have been listed in a table using the List Results option, then you can select certain results in the list by pressing Click + CTRL. Alternately, you can select a range of results in the list by pressing Shift + Click.

9 Click Close.

Toolbar: Text Formatting 

Command entry: FIND

Change Text Scale and Justification

Several commands are available for changing the scale of one or more text and attribute objects, or their insertion points, simultaneously without changing the location of the objects.

You can change the scale of one or more text objects, attributes, and attribute definitions, or their insertion points, simultaneously without changing the location of the objects.

Change the Scale of Multiple Text Objects

A drawing may contain hundreds of text objects that need to be scaled, and it would be tedious to scale them individually. Use SCALETEXT to change the scale of one or more text objects such as text, multiline text, and attributes. You can specify a relative scale factor or an absolute text height, or you can scale selected text to match the height of existing text. Each text object is scaled using the same scale factor, and it maintains its current location.

Convert Text Height Between Model Space and Paper Space

The SPACETRANS command calculates equivalent lengths between model space units and paper space units. By using SPACETRANS transparently, you can provide commands with distance or length values relative to another space. For example, you may want to create a text object in model space that matches the height of other text in a layout. From model space, you could enter

Command: **text**
Specify start point of text or [Justify/Style]: **1,3**
Specify height <0.375>: '**spacetrans**
>>Specify paper space distance <1.000>: **3/8**

Resuming TEXT command
Specify height <0.375>: **1.173**

When the command is complete, a text object is created in model space with a height of 1.173, which appears as 3/8 when viewed from a layout.

NOTE The SPACETRANS command is not available from the Model tab or in a perspective view.

For more information about entering commands transparently, see Enter Commands on the Command Line on page 39.

Change the Justification of Text Objects Without Changing Their Location

Use JUSTIFYTEXT to redefine the insertion point of text without moving the text. For example, a table or schedule may contain text that is located correctly but each text object in the table should be right-justified instead of left-justified for future entries or modifications.

To scale multiline text objects without changing their locations

1 Click Annotate tab ➤ Text panel ➤ Scale. 

2 Select one or more multiline text objects and press ENTER.

3 Specify one of the justification options or press ENTER to accept the existing text justifications.

4 Enter **s** and enter the scale factor to be applied to each mtxt object.

Command entry: SCALETEXT

Quick Reference

Commands

JUSTIFYTEXT

Changes the justification point of selected text objects without changing their locations.

PROPERTIES

Controls properties of existing objects.

SCALETEXT

Enlarges or reduces selected text objects without changing their locations.

SPACETRANS

Calculates equivalent model space and paper space lengths in a layout.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Check Spelling

You can check the spelling of all text as it is entered in your drawing. You can also specify the specific language dictionary that is used and customize and manage multiple custom spelling dictionaries.

You can check the spelling of all text objects in your drawing, including

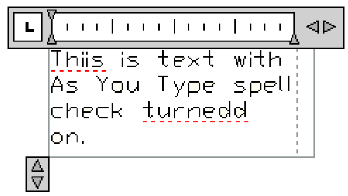
- Single and multiline text
- Dimension text
- Multileader text
- Text within block attributes
- Text within xrefs

With Check Spelling, your drawing or the areas of your drawing's text that you specify are searched for misspelled words. If a misspelled word is identified, the word is highlighted and the drawing area zooms to that word in a scale that is easy to read.

Check Spelling As You Type

By default, you can check spelling as you enter text in the In-Place Text Editor. Any word you enter is checked for spelling errors when it is completed. A word is considered completed when one of the following actions are taken:

- Pressing SPACEBAR or ENTER
- Moving the cursor to another position within the In-Place Text Editor.



Misspelled words are underlined with a red dotted line

Any word not found in the current dictionary is underlined as misspelled. Spelling suggestions are displayed when you right-click the underlined word.

Switch Dictionaries

The Check Spelling feature contains several main dictionaries, which are available in different languages. You can also create any number of custom dictionaries and switch between them as needed.

During a spelling check, the words in the drawing are matched to the words in the current main dictionary. Any words you add are stored in the custom dictionary that is current at the time of the spelling check. For example, you can add proper names so they are no longer identified as misspelled words.

NOTE By default AutoCAD provides you with one sample custom dictionary that contains words such as AutoCAD and Autodesk.

To check spelling in another language, change to a different main dictionary.

You can change dictionaries in the Dictionaries dialog box or by specifying the dictionary name in the DCTMAIN or DCTCUST system variable. For a list of the main dictionary file names, see DCTMAIN.

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.

Create and Edit Custom Dictionaries

A custom dictionary is a list of spelling exceptions that you have identified. The files that contain them have a *.cus* file extension. You can use any ASCII text editor to add or delete words, or combine several dictionaries.

To check spelling

1 Click Annotate tab ➤ Text panel ➤ Check Spelling.

2 Click an option of where you want to check. Click Start. If no misspelled words are found, a message is displayed. If a misspelling is found, the Check Spelling dialog box identifies the misspelled word. The word is highlighted and zoomed to in the drawing area.

3 Do *one* of the following:

- To correct a word, select an alternate word from the Suggestions list or type a word in the Suggestions box. Click Change or Change All.
- To leave a word unchanged, click Ignore or Ignore All.
- To leave a word unchanged and add it to the dictionary, click Add to Dictionary.

4 Repeat step 3 for each misspelled word. Click Close to exit.

NOTE Click Undo to reverse the preceding Check Spelling action or series of actions in the Check Spelling dialog box.

Toolbar: Text
Command entry: SPELL

To check spelling in a block attribute

1 Click Annotate tab ➤ Text panel ➤ Check Spelling.

2 In the Check Spelling dialog box, click Settings.

3 In the Check Spelling Settings dialog box, click Block Attributes. Click OK.

4 In the Check Spelling dialog box, click Start.

5 Do *one* of the following:

- To correct a word, select an alternate word from the Suggestions list or type a word in the Suggestions box. Click Change or Change All.
- To leave a word unchanged, click Ignore or Ignore All.
- To leave a word unchanged and add it to the dictionary, click Add to Dictionary.

6 Repeat step 5 for each misspelled word. Click Close to exit.

Toolbar: Text

Command entry: SPELL

To switch dictionaries while checking spelling

1 Click Annotate tab ➤ Text panel ➤ Check Spelling.

2 In the Check Spelling dialog box, click Dictionaries.

3 Do *one* of the following:

- To change the main dictionary, select a dictionary from the Current Main Dictionary list.
- To change the custom dictionary, select a dictionary under Current Custom Dictionary.

4 Click Close.

Command entry: SPELL

To add a custom dictionary or word list

1 Click Annotate tab ➤ Text panel ➤ Check Spelling.

2 In the Check Spelling dialog box, click Dictionaries.

3 In the Dictionaries dialog box, in the Current Custom dictionary list, select Manage Custom Dictionaries.

4. In the Custom Dictionaries list, click Add and browse to the dictionary's location. To create a new custom dictionary, click New and enter the dictionary's name. The name must contain the *.cus* extension.
5. Click OK. The newly selected dictionary is highlighted as the current custom dictionary.
6. If you would like to import a word list into your custom dictionary, click Import.
7. Click OK.

Command entry: SPELL

Quick Reference

Commands

SPELL

Checks spelling in a drawing.

System Variables

DCTCUST

Displays the path and file name of the current custom spelling dictionary.

DCTMAIN

Displays the three letter keyword for the current main spelling dictionary.

Utilities

No entries

Command Modifiers

No entries

Use an Alternate Text Editor

The default text editor is either the MTEXT ribbon contextual tab (if the ribbon is active) or the In-Place Text Editor (if the ribbon is not active), but you can elect to use any alternate editor that saves files in ASCII format.

Overview of Using an Alternate Text Editor

You can use any text editor, such as Microsoft Notepad, that saves files in ASCII format.

You can elect to use an alternate editor by specifying the editor with the MTEXTED system variable.

If you use an alternate text editor for multiline text, you specify the properties of the multiline text object at the Command prompt first. Then the text editor opens for entering text. When you close the text editor, the text is inserted within the width limit you specified.

If you use an alternate editor, you must enter special codes to apply formatting.

To edit text using an alternate text editor, use the same format codes. To avoid losing format information when you make changes to the text, use the same text editor you used to create the text.

Quick Reference

Commands

MTEXT

Creates a multiline text object.

OPTIONS

Customizes the program settings.

System Variables

MTEXTED

Sets the application for editing multiline text objects.

Utilities

No entries

Command Modifiers

No entries

Format Multiline Text in an Alternate Text Editor

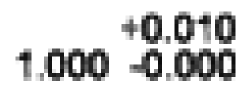
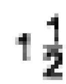
If you use an alternate text editor, you apply formatting by entering format codes.

You can underline text, add a line over text, and create stacked text. You can also change color, font, and text height. You can change the spaces between text characters or increase the width of the characters themselves. To apply formatting, use the format codes shown in the following table.

Format codes for paragraphs

Format code	Purpose	Enter this ...	To produce this ...
\0...\o	Turns overline on and off	Autodesk \OAutoCAD\o	Autodesk AutoCAD
\L...\l	Turns underline on and off	Autodesk \LAutoCAD\l	Autodesk <u>AutoCAD</u>
\~	Inserts a nonbreaking space	Autodesk AutoCAD\~LT	Autodesk AutoCAD LT
\\	Inserts a backslash	Autodesk \\AutoCAD	Autodesk \AutoCAD
\{...\}	Inserts an opening and closing brace	Autodesk \{AutoCAD\}	Autodesk {AutoCAD}
\C*value;*	Changes to the specified color	Autodesk \C2;AutoCAD	Autodesk AutoCAD
File name;	Changes to the specified font file	Autodesk \Ftimes; AutoCAD	Autodesk AutoCAD

Format codes for paragraphs

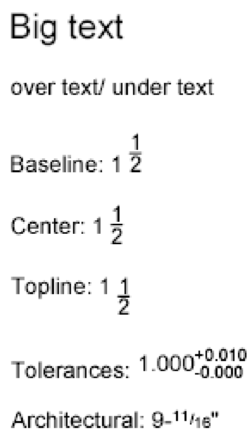
Format code	Purpose	Enter this …	To produce this …
\H*value*;	Changes to the text height specified in drawing units	Autodesk \H2;AutoCAD	
\H*valuex*;	Changes the text height to a multiple of the current text height	Autodesk \H3x;AutoCAD	
\S...^...;	Stacks the subsequent text at the /, #, or ^ symbol	1.000\S+0.010^-0.000;	
\T*value*;	Adjusts the space between characters. Valid values range from a minimum of .75 to 4 times the original spacing between characters.	\T2;Autodesk	
\Q*angle*;	Changes obliquing angle	\Q20;Autodesk	
\W*value*;	Changes width factor to produce wide text	\W2;Autodesk	
\A	Sets the alignment value; valid values: 0, 1, 2 (bottom, center, top)	\A1;1\S1/2	
\P	Ends paragraph	Autodesk\PAutoCAD	

Braces can be nested up to eight levels deep.

You can also use control codes to add special characters, such as tolerance or dimensioning symbols. See MTEXT.

Example: Formatting Text in an Alternate Text Editor

This example describes how the text in the following illustration was created.



Each line below was entered in an alternate text editor:

```
{{\H1.5x; Big text} \A2; over text\A1;/\A0; under text}\P
{\A0;Baseline: 1 \S1/2;}\P
{\A1;Center: 1 \S1/2;}\P
{\A2;Topline: 1 \S1/2;}\P
{Tolerances: \A1;1.000\H.75x;\S+0.010^-0.000;}\P
{Architectural: 9-{\H.666x;\A2;11\A1;/\A0;16}\A2;"}\P
```

To specify an alternate text editor

1 At the Command prompt, enter **mtexted**.

2 At the prompt do *one* of the following:

- Enter the path and name of the executable file for the ASCII text editor that you want to use to create or edit multiline text.
- Enter **internal** to restore the text editor.

To create multiline text in an alternate text editor

1. To specify a text editor, at the Command prompt, enter **mtexted**. Then enter the path of the editor you want to use.
2. Click Home tab ➤ Annotation panel ➤ Multiline Text.
3. Specify the first corner of the multiline text boundary rectangle.
4. Specify the opposite corner of the multiline text boundary rectangle.
5. In the text editor, enter the text. Enter **\P** to end a paragraph and start a new paragraph on the next line. (Be sure to capitalize the P.)
6. When your text entry is complete, save the changes and exit the text editor.

Toolbar: Draw
Command entry: MTEXT

Quick Reference

Commands

MTEXT

Creates a multiline text object.

OPTIONS

Customizes the program settings.

System Variables

MTEXTED

Sets the application for editing multiline text objects.

Utilities

No entries

Command Modifiers

No entries

Tables

26

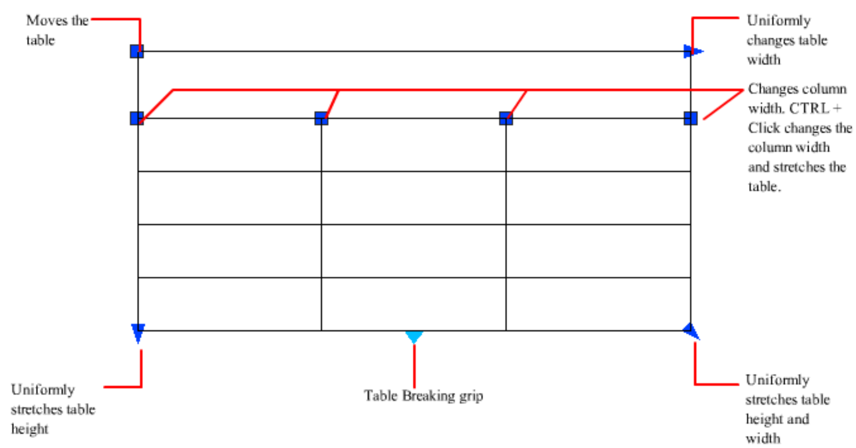
A table is a rectangular array of cells that contain annotation, primarily text but also blocks. Tables appear in many different forms on many of the sheets that make up drawing sets. In the AEC industry, tables are often referred to as “schedules” and contain information about the materials needed for the construction of the building being designed. In the manufacturing industry, they are often referred to as “BOM” (bills of materials).

The table object creates a table of any size that can be used for any purpose, including as a list or index to a set of drawing sheets to be published.

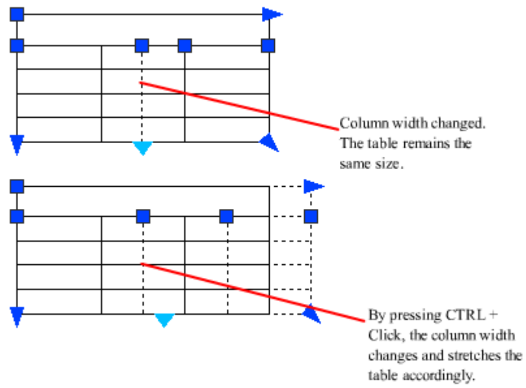
Create and Modify Tables

A table is an object that contains data in rows and columns. A table object can be created from an empty table or table style. A table can also be linked to data in a Microsoft Excel spreadsheet.

After the table has been created, you can click any gridline on the table to select it and then modify it by using the Properties palette or grips.



When you change the height or width of the table, only the row on page 1949 or column on page 1928 adjacent to the grip you have selected will change. The table will maintain its height or width. To change the size of the table proportionally to the size of the row or column you are editing, press CTRL while using a column grip.

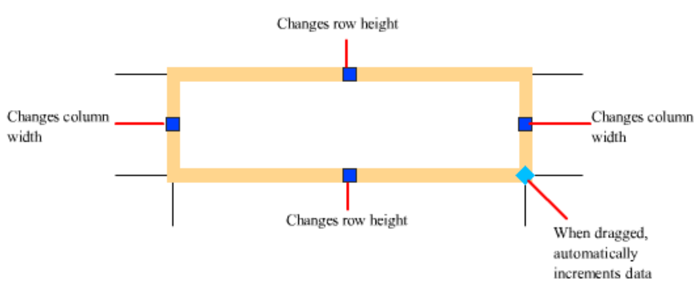


Break Tables into Multiple Parts

A table with a large amount of data can be broken into primary and secondary table fragments. Use the table breaking grips found at the bottom of your table to make a table span multiple columns in your drawing or to manipulate the different table parts you have already created.

Modify a Table Cell

You can click inside a cell to select it. Grips are displayed in the middle of the cell borders. Click inside another cell to move selection to that cell. Drag the grips on a cell to make the cell and its column or row larger or smaller.



NOTE When a cell is selected, press F2 to edit the cell text.

To select more than one cell, click and drag over several cells. You can also hold down SHIFT and click inside another cell to select those two cells and all the cells between them.

When you click inside a table cell when the ribbon is active, the Table ribbon contextual tab is displayed. If the ribbon is not active, the Table toolbar is displayed. From here, you can

- Edit rows and columns
- Merge and unmerge cells
- Alter the appearance of cell borders
- Edit data formatting and alignment
- Lock and unlock cells from editing
- Insert blocks, fields, and formulas
- Create and edit cell styles
- Link the table to external data

With a cell selected, you can also right-click and use the options on the shortcut menu to insert or delete columns and rows, combine adjacent cells, or make other changes. When cells are selected, you can use CTRL+Y to repeat the last action.

NOTE Using CTRL+Y to repeat the last action only repeats actions executed through the shortcut menu, the Table ribbon contextual tab, or the Table toolbar.

Add a Table to a Tool Palette

When you add a table to a tool palette, the table properties (for example, table style and number of rows and columns) and the cell property overrides (for example, alignment and border lineweight) are stored in the tool definition. The text, block content, and character formatting are also stored in the tool definition.

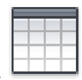
Customize Display of Column Letters and Row Numbers

By default, the In-Place Text Editor displays column letters and row numbers when a table cell is selected for editing. Use the TABLEINDICATOR system variable to turn this display on and off. To set a new background color, select a table, right-click, and click Table Indicator Color on the shortcut menu. The text color, size, and style and the line color are controlled by the settings for column heads in the current table style.

See also:

- Add Text and Blocks to Tables on page 1606

To create an empty table

1 Click Home tab ➤ Annotation panel ➤ Insert Table. 

2 In the Insert Table dialog box, select a table style from the list, or click the button to the right of the drop-down menu to create a new table style.

3 Click Start from Empty Table.

4 Insert the table in the drawing by doing one of the following:
   - Specify an insertion point for the table.
   - Specify a window for the table.

5 Set the number of columns and the column width.

   If you used the window insertion method, you can select the number of columns or the column width, but not both.

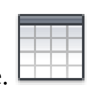
6 Set the number of rows and the row height.

   If you used the window insertion method, the number of rows is determined by the size of the window you specified and the row height.

7 Click OK.

Command entry: TABLE

To create a table from a linked spreadsheet

1 Click Home tab ➤ Annotation panel ➤ Insert Table. 

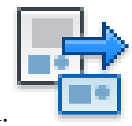
2 Click From a Data Link

3 Select an established data link from the drop-down menu, or click the [...] button to create a new data link using the Data Link Manager.

4 Click OK to specify an insertion point in the drawing for the table.

Command entry: TABLE

To create a table from a data extraction



1 Click Insert tab ➤ Linking & Extraction panel ➤ Extract Data.

2 In the Data Extraction wizard, on the Begin page, click Create a New Data Extraction. If you want to use a template (DXE or BLK) file, click Use Previous Extraction as a Template. Click Next.

3 In the Save Data Extraction As dialog box, specify a file name for the data extraction file. Click Save.

4 On the Define Data Source page, specify the drawings or folders from which to extract data. Click Next.

5 On the Select Objects page, select the objects from which to extract data. Click Next.

6 On the Select Properties page, select the properties from which to extract data. Click Next.

7 On the Refine Data page, organize the columns if necessary. Click Next.

8 On the Choose Output page, click Insert Data Extraction Table into Drawing to create a data extraction table. Click Next.

9 On the Table Styles page, choose a table style if one is defined in the current drawing, or table if defined in a table style. Enter a title for the table if necessary. Click Next.

10 In the Finish page, click Finish.

11 Click an insertion point in the drawing to create the table.

Toolbar: Modify II 
Command entry: DATAEXTRACTION

To create a tool from a table in the current drawing

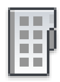
1 In the current drawing, select the table.

2 Using the right mouse button, drag the table to a tool palette and, without releasing the mouse button, move the cursor to the place on the tool palette where you want the tool.

   You can switch to a different tab by hovering over the tab for a few seconds. The black line indicates where the tool will be located.

3 Release the mouse button.

   NOTE All formatting, table properties, and cell properties are saved in the tool palette tool, as well as text and block content.

Toolbar: Standard 
Command entry: TOOLPALETTES

To lock and unlock cells

1 Select one or more cells in a table that you want to lock or unlock using one of the following methods:
   - Click inside a cell.
   - Hold down SHIFT and click inside another cell to select those two cells and all the cells between them.
   - Click inside the selected cell, drag to the cells you want to select, and release.

2 Use one of the following options:
   - **To unlock a cell or cells.** On the Table ribbon contextual tab or Table toolbar, click Locking ➤ Unlocked
   - **To lock a cell or cells.** On the Table ribbon contextual tab or Table toolbar, click Locking ➤ Data and Format Locked

Command entry: TABLEDIT
Shortcut menu: Select and right-click a cell or range of cells. Click Locking.

To use grips to modify a table

1 Click a gridline to select the table.

2 Use one of the following grips:

 - **Upper-left grip**. Moves the table.
 - **Upper-right grip**. Changes the width of the table and all columns proportionally.
 - **Lower-left grip**. Changes the height of the table and all rows proportionally.
 - **Lower-right grip**. Changes both the height and the width of the table and changes the columns and rows proportionally.
 - **Column grip** (at the top of the column head row). Widens or narrows adjacent columns without changing the width of the table.
 - **CTRL+ a column grip**. Changes the width of the column to the left of the grip and widens or narrows the table to accommodate the change.

 The minimum column width is the width of a single character. The minimum row height for an empty table is the height of a line of text plus the cell margins.

3 Press ESC to remove selection.

To use grips to modify cells in a table

1 Select one or more cells in a table that you want to change using one of the following methods:

 - Click inside a cell.
 - Hold down SHIFT and click inside another cell to select those two cells and all the cells between them.
 - Click inside the selected cell, drag to the cells you want to select, and release.

2 To change the row height of the selected cell, drag the top or bottom grip.

 If more than one cell is selected, the row height changes equally for each row.

3 To change the column width of the selected cell, drag the left or right grip.

If more than one cell is selected, the column width changes equally for each column.

4 To merge selected cells, click Merge Cells on either the Table ribbon contextual tab or Table toolbar.

If cells in more than one row or column are selected, you can merge by row or by column.

5 Press ESC to remove selection.

To use the Properties palette to modify a table

1 Click a gridline to select the table.

2 Click View tab ➤ Palettes panel ➤ Properties. 

3 In the Properties palette, click the value you want to change and enter or select a new value.

The property is changed in the selected table.

4 Move the cursor outside the Properties palette, and press ESC to remove selection.

Toolbar: Properties 
Command entry: PROPERTIES
Shortcut menu: Select and right-click a cell or range of cells. Click Properties.

To break a table into multiple parts using grips

1 Click a gridline to select the table.

2 Click the triangular grip at the bottom center gridline of the table.

- When the triangle is pointed down, then table breaking is inactive. New rows will be added to the bottom of the table.
- When the triangle is pointed up, then table breaking is active. The current position of the bottom of the table is the maximum height of the table. Any new rows will be added to a second table part to the right of the master table part.

To change column width or row height in a table

1 Click inside a table cell in the column or row you want to change.

Hold down SHIFT and click inside another cell to select those two cells and all the cells between them.

2 Right-click. Click Properties.

3 In the Properties palette, under Cell, click the cell width or the cell height value and enter a new value.

4 Press ESC to remove selection.

To add columns or rows to a table

1 Click inside a table cell where you want to add a column or a row.

You can select more than one cell to add more than one column or row.

2 On either the Table ribbon contextual tab or Table toolbar, select one of the following options:

- **Insert Row Above**. Inserts a row above the selected cell.
- **Insert Row Below**. Inserts a row below the selected cell.
- **Insert Column Left**. Inserts a column to the left of the selected cell.
- **Insert Column Right**. Inserts a column to the right of the selected cell.

NOTE The cell style of the new column or row will be the same as that of the originally selected column or row. To change the cell style, right-click the cell or cells and click Cell Style.

3 Press ESC to remove selection.

Shortcut menu: Select and right-click a cell or range of cells. Click Rows or Columns and select an insertion method.

To merge cells in a table

1 Select the cells in a table that you want to merge using one of the following methods:

- Select a cell, and hold down SHIFT and click inside another cell to select those two cells and all the cells between them.

- Click inside a selected cell, drag to the cells you want to select, and release.

The resulting merged cell must be rectangular.

2 Click Merge Cells on either the Table ribbon contextual tab or Table toolbar. If you want to create more than one merged cell, use one of the following options:
   - **All:** Merges all cells in a rectangular selected range.
   - **By Row:** Merges the cells horizontally by removing the vertical gridlines and leaving the horizontal gridlines intact.
   - **By Column:** Merges the cells vertically by removing the horizontal gridlines and leaving the vertical gridlines intact.

3 Start typing to enter text in the new merged cell, or press ESC to remove selection.

Shortcut menu: Select and right-click a cell or range of cells. Click Merge and select a merging method.

To delete columns or rows in a table

1 Click inside a table cell in the column or row that you want to delete.

   Hold down SHIFT and click inside another cell to select those two cells and all the cells between them.

2 To delete rows, select Delete Rows on either the Table ribbon contextual tab or the Table toolbar. To delete columns, select Delete Columns on either the Table ribbon contextual tab or the Table toolbar.

   NOTE Rows and columns that contain a portion of a data link cannot be deleted.

3 Press ESC to remove selection.

Shortcut menu: Select and right-click a cell or range of cells. Click Columns ➤ Delete, or Rows ➤ Delete.

Quick Reference

Commands

FIELD

Creates a multiline text object with a field that can be updated automatically as the field value changes.

FIND

Finds the text that you specify, and can optionally replace it with other text.

INSERT

Inserts a block or drawing into the current drawing.

LIST

Displays property data for selected objects.

MTEXT

Creates a multiline text object.

SPELL

Checks spelling in a drawing.

TABLE

Creates an empty table object.

TABLEDIT

Edits text in a table cell.

TABLEEXPORT

Exports data from a table object in CSV file format.

TABLESTYLE

Creates, modifies, or specifies table styles.

UPDATEFIELD

Manually updates fields in selected objects in the drawing.

System Variables

CTABLESTYLE

Sets the name of the current table style.

FIELDDISPLAY

Controls whether fields are displayed with a gray background.

FIELDEVAL

Controls how fields are updated.

TABLETOOLBAR

Controls the display of the Table toolbar.

Utilities

No entries

Command Modifiers

No entries

Link a Table to External Data

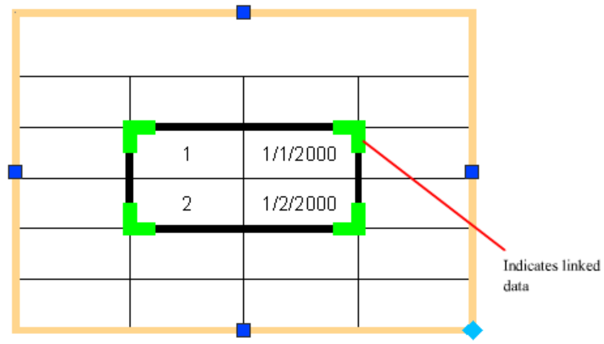
A table can be linked to data in a Microsoft Excel (**XLS**, **.XLSX**, or **CSV**) file. You can link to an entire spreadsheet, individual row, column, cell, or cell range in Excel.

NOTE Microsoft Excel must be installed to use Microsoft Excel data links. To link to the XLSX filetype, Microsoft Excel 2007 must be installed.

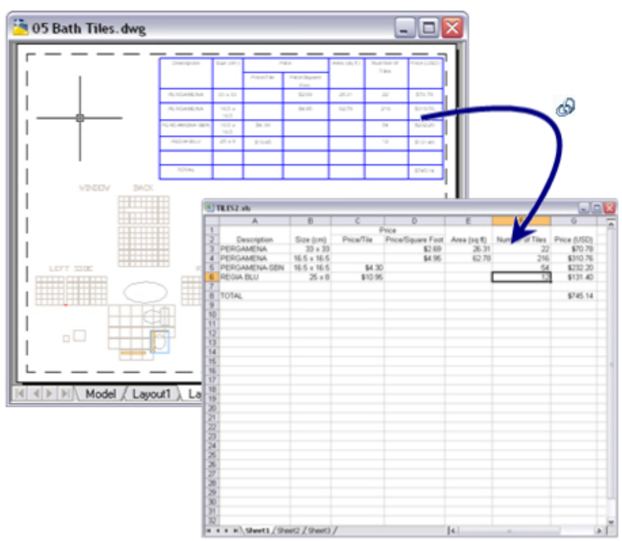
You can bring data from Microsoft Excel into a table in the following three ways:

- As formulas with supported data formats attached.
- As calculated data from formulas calculated in Excel (supported data formats not attached).
- As calculated data from formulas calculated in Excel (with data formats attached).

A table that contains data links displays indicators around linked cells. If you hover your mouse cursor over the data link, information about the data link is displayed.



If a linked spreadsheet has been changed, such as adding a row or column, the table in your drawing can be updated accordingly using the DATALINKUPDATEcommand. Likewise, if a change is made to a table in your drawing, then you can update the linked spreadsheet using the same command.



By default, a data link is locked from editing to prevent undesired changes to the linked spreadsheet. You can lock cells from data changes, format changes, or both. To unlock a data link, click Locking on either the Table ribbon contextual tab or the Table toolbar.

See also:

- Extract Data from Drawings and Spreadsheets

To link to a table in an external spreadsheet

1 Click Annotate tab ➤ Tables panel ➤ Data Link.

2 In the Data Link Manager tree view, click Create a New Excel Data Link.

3 In the Enter Data Link Name dialog box, enter a name for the data link. Click OK.

4 Click the [...] button to browse for the XLS or CSV file to be linked to.

NOTE To link to the XLSX filetype, Microsoft Excel 2007 must be installed.

5 In the New Excel Data Link dialog box, select a link option (whole sheet, range, or Excel named range). Click OK.

6 Select the new data link in the Data Link Manager tree view. Click OK.

Command entry: DATALINK

To remove a link to an external spreadsheet

1 Click inside a cell within the data linked table to select the cell.

2 Right-click and click Data Links ➤ Detach Data Link.

Command entry: DATALINK

To set up a named range in Microsoft Excel

1 In Microsoft Excel, open the workbook or spreadsheet that you want to access.

2 Select a range of cells to function as a linked range.

3 In the Name Box, enter a name for the range of cells, then press ENTER.

4 Repeat steps 2 and 3, if desired, to specify additional linked ranges.

5 On the File menu (Microsoft Excel), choose Save.

To link a table to a named range in Microsoft Excel

1 In your table, select the table cells to link.

2 On either the Table ribbon contextual tab or Table toolbar, click Link Cell.
3 In the Data Link Manager tree view, select Click to Create a New Excel Data Link.
4 In the Enter Data Link Name dialog box, enter a name for the data link. Click OK.
5 Click the [...] button to browse for the XLS or CSV file to link.
6 In the New Excel Data Link dialog box, select Link to a Named Range. Click OK.
7 Select the new data link in the Data Link Manager tree view. Click OK.

Command entry: DATALINK

To link a table to cells in Microsoft Excel

1 In your table, select the table cells to link.
2 On the Table toolbar, click Link Cell.
3 In the Data Link Manager tree view, select Click to Create a New Excel Data Link.
4 In the Enter Data Link Name dialog box, enter a name for the data link. Click OK.
5 Click the [...] button to browse for the .xls or .csv file to be linked to.
6 In the New Excel Data Link dialog box, select Link to Range. Enter a valid range from the Excel spreadsheet (for example, A1:D17). Click OK.
7 Select the new data link in the Data Link Manager tree view. Click OK.

Command entry: DATALINK

To update changed data between a drawing and Microsoft Excel

1 Click Annotate tab ➤ Tables panel ➤ Data Link. 
2 On the Data Link flyout, click Update Data Links.

Command entry: DATALINK

To open an external spreadsheet from a data link

1 Select any cell in the linked table or range of linked cells.

2 Right-click and click Data Links ➤ Open Data Link File.

Quick Reference

Commands

DATALINK

The Data Link dialog box is displayed.

DATALINKUPDATE

Updates data to or from an established external data link.

System Variables

DATALINKNOTIFY

Controls the notification for updated or missing data links.

Utilities

No entries

Command Modifiers

No entries

Work with Table Styles

The appearance of the table is controlled by its table style. You can use the default table style, STANDARD, or create your own table styles.

When you create a new table style, you can specify a starting table. A starting table is a table in your drawing that is used as an example for formatting the new table style. Once a table is selected, you can specify the structure and contents to copy from that table to the table style.

Cell styles can be created and applied to a table style upon insertion of a new table. A table style can specify different cell styles in each type of row to display a different justification and appearance for the text and gridlines. These cell styles are specified upon insertion of a table. The STANDARD table style, for example, contains a cell style consisting of merged cells with text that is centered. This cell style, named Title, can be specified as the first row cell of the table. This creates a title row at the top of the new table.

The table can read from top to bottom or from the bottom up. The number of columns and rows is almost unlimited.

The border properties in a table's cell style control the display of the gridlines that divide the table into cells. The borders of the title row, the column heads row, and the data rows can have different lineweight and color and can be displayed or not displayed. The Cell Style preview image in the bottom right corner of the Table Style dialog box updates as you select border options.

The appearance of text in the cells of the table is controlled by the text style that is specified in the current cell style. You can use any text style in the drawing or create a new one. You can also use DesignCenter to copy table styles from other drawings.

You can define the data and formatting for any cell style within a table style. You can also overwrite the data and formatting for specific cells. For example, you could set the formatting for all column heading rows to display text in uppercase, and then select a single table cell to display text in lowercase. The type of data you display in a row and the formatting for that data type is controlled by the formatting options you select in the Table Cell Format dialog box.

See also:

- Work with Text Styles on page 1544

To define or modify a table style

1 Click Home tab ➤ Annotation panel ➤ Table Style. 

2 In the Table Style dialog box, click New.

3 In the Create New Table Style dialog box, enter a name for the new table style.

4 In the Start With drop-down list, select a table style to provide default settings for the new table style. Click Continue

5 In the New Table Style dialog box, click the Select Table button to select a table in your drawing to which to apply the new table style settings.

6 In the Table Direction drop-down list, select Down or Up. Up creates a table that reads from the bottom up; the title row and the column heads are at the bottom of the table.

7 In the Cell Styles drop-down list, select a cell style to apply to the table, or create a new cell style by clicking the button to the right of the drop-down list.

8 On the General tab, select or clear the following options for the current cell style:

- **Fill Color**. Specifies the fill color. Select None or a background color, or click Select Color to display the Select Color dialog box.
- **Alignment**. Specifies an alignment for cell contents. Center refers to horizontal alignment; Middle refers to vertical alignment.
- **Format**. Sets the data type and formatting for rows in a table. Click the [...] button to display the Table Cell Format dialog box, where you can further define formatting options.
- **Type**. Specifies the cell style as either a label or data, which is used when inserting default text in a table style containing a starting table. Also used when creating a table tool on the tool palette.
- **Margins - Horizontal**. Sets the distance between the text or block in the cell and the left and right cell borders.
- **Margins - 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.

9 On the Text tab, select or clear the following options for the current cell style:

- **Text Style**. Specifies the text style. Select a text style, or click the [...] button to open the Text Style dialog box and create a new text style.
- **Text Height**. Specifies the text height. Enter a height for the text. This option is available only when the selected text style has a text height of 0. (The default text style, STANDARD, has a 0 text height.) If the selected text style specifies a fixed text height, this option is unavailable.
- **Text Color**. Specifies the text color. Select a color, or click Select Color to display the Select Color dialog box.

- **Text Angle**. Sets the text angle. The default text angle is 0 degrees. You can enter any angle between -359 and +359 degrees.

10 Use the Borders tab to control the appearance of the table gridlines for the current cell style. Specify the following options:

- **Lineweight**. Sets the lineweight to use for borders that are displayed. If you use a heavy lineweight, you may have to change the cell margins so that the text can be seen.
- **Linetype**. Sets the linetype to apply 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**. Specifies a color to use for borders that are displayed. Click Select Color to display the Select Color dialog box.
- **Double Line**. Specifies that the selected borders have a double linetype. Change the spacing between the lines by entering a value in the Spacing box.
- **Border display buttons**. Applies the selected border options. Click a button to apply the selected border options to all borders of the cell, the outside border, the inside borders, the bottom border, the left border, the top border, the right border, or no borders. The preview in the dialog box updates to show the effect.

11 Click OK.

Toolbar: Styles 
Command entry: TABLESTYLE

To define or modify a cell style

1 Click Home tab ➤ Annotation panel ➤ Table Style. 

2 Select the table style that contains the cell style you want to modify, or click New to create a new table style.

3 In the Table Style dialog box, in the Cell Styles drop-down list, select a cell style to modify, or create a new cell style by clicking the button to the right of the drop-down list.

4 On the General tab, select or clear the following options for the current cell style:

- **Fill Color**. Specifies the fill color. Select None or a background color, or click Select Color to display the Select Color dialog box.
- **Alignment**. Specifies an alignment for cell contents. Center refers to horizontal alignment; Middle refers to vertical alignment.
- **Format**. Set the data type and formatting for rows in a table. Click the [...] button to display the Table Cell Format dialog box, where you can further define formatting options.
- **Type**. Specifies the cell style as either a label or data, which is used when inserting default text in a table style containing a starting table. Also used when creating a table tool on the tool palette.
- **Margins - Horizontal**. Sets the distance between the text or block in the cell and the left and right cell borders.
- **Margins - 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.

5 On the Text tab, select or clear the following options for the current cell style:

- **Text Style**. Specifies the text style. Select a text style, or click the [...] button to open the Text Style dialog box and create a new text style.
- **Text Height**. Specifies the text height. Enter a height for the text. This option is available only when the selected text style has a text height of 0. (The default text style, STANDARD, has a 0 text height.) If the selected text style specifies a fixed text height, this option is unavailable.
- **Text Color**. Specifies the text color. Select a color, or click Select Color to display the Select Color dialog box.
- **Text Angle**. Sets the text angle. The default text angle is 0 degrees. You can enter any angle between -359 and +359 degrees.

6 Use the Borders tab to control the appearance of the table gridlines for the current cell style. Specify the following options:

- **Lineweight**. Sets the lineweight to use for borders that are displayed. If you use a heavy lineweight, you may have to change the cell margins so that the text can be seen.
- **Linetype**. Sets the linetype to apply 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**. Specifies a color to use for borders that are displayed. Click Select Color to display the Select Color dialog box.
- **Double Line**. Specifies that the selected borders will have a double linetype. Change the spacing between the lines by entering a value in the Spacing box.
- **Border display buttons**. Applies the selected border options. Click a button to apply the selected border options to all borders of the cell, the outside border, the inside borders, the bottom border, the left border, the top border, the right border, or no borders. The preview in the dialog box updates to show the effect.

7 Click OK.

Toolbar: Styles
Command entry: TABLESTYLE

To create a table style from an existing table

1 Click a gridline to select the table.

2 Right-click and click Table Style ➤ Save as New Table Style.

Toolbar: Styles
Command entry: TABLESTYLE

To create a cell style from an existing cell

1 Click inside the cell to create a cell style from.

2 Right-click and click Cell Style ➤ Save as New Cell Style.

Toolbar: Styles
Command entry: TABLESTYLE

To apply a new table style to a table

1 Click a gridline to select the table.

2 Right-click and select Table Style.

3 On the Table Style flyout, select a table style from the list.
The new table style is applied to the table.

NOTE If the previous table style had a title row and the new one does not, the title text is placed in the first cell of the table, and the other cells in the first row are left blank.

---

4 Press ESC twice to remove selection.

Toolbar: Styles
Command entry: TABLESTYLE

To change the table style that is applied to new tables

1 Click Home tab ➤ Annotation panel ➤ Table Style.

2 In the Table Style dialog box, select a table style. Click Set Current.

3 Click Close.
The current table style is applied to new tables that you create.

Toolbar: Styles
Command entry: TABLESTYLE

Quick Reference

Commands

FIND

Finds the text that you specify, and can optionally replace it with other text.

INSERT

Inserts a block or drawing into the current drawing.

LIST

Displays property data for selected objects.

MTEXT

Creates a multiline text object.

SPELL

Checks spelling in a drawing.

TABLE

Creates an empty table object.

TABLEDIT

Edits text in a table cell.

TABLEEXPORT

Exports data from a table object in CSV file format.

TABLESTYLE

Creates, modifies, or specifies table styles.

UPDATEFIELD

Manually updates fields in selected objects in the drawing.

System Variables

CTABLESTYLE

Sets the name of the current table style.

FIELDDISPLAY

Controls whether fields are displayed with a gray background.

FIELDEVAL

Controls how fields are updated.

Utilities

No entries

Command Modifiers

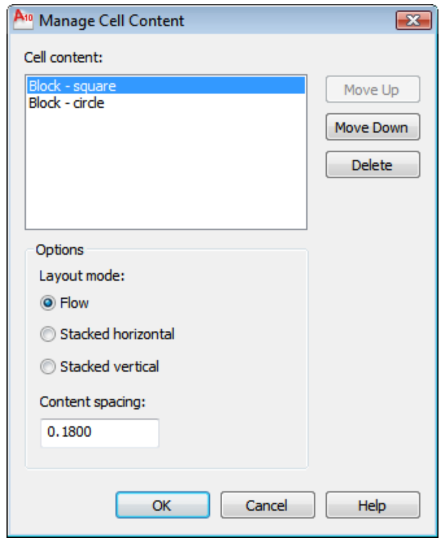
No entries

Add Text and Blocks to Tables

Table cell data can include text and multiple blocks.

When a table is created, the first cell is highlighted, the Text Formatting toolbar is displayed, and you can begin entering text. The row height of the cell increases to accommodate the number of lines of text. To move to the next cell, press TAB, or use the arrow keys to move left, right, up, and down. You can quickly edit cell text by pressing F2 in a selected cell.

When you insert a block into a table cell, either the block can be automatically fit to the size of the cell, or the cell can adjust to accommodate the size of the block. Blocks can be inserted from the Table toolbar, or from the shortcut menu. Multiple blocks can be inserted in a table cell. If there is more than one block in a table cell, use the Manage Cell Content dialog box to customize the way the cell content is displayed.



Inside the cell, the arrow keys move the cursor. Use the Table toolbar and the shortcut menu to format text, import text, or make other changes to the text in the cell.

See also:

- Use Fields in Text on page 1532
- Create Multiline Text on page 1481

To enter text in a table

1. Click inside a table cell, and start to enter text.

   The Text Formatting toolbar is displayed.

2. Use the arrow keys to move the cursor in text inside a cell.
3. To create a line break within a cell, press ALT+ENTER .
4. To override the text style specified in the table style, click the arrow next to the Text Style control on the toolbar and select a new text style.

   The text style you select is applied to the text in the cell and any new text that you enter in the cell.

5 To override the formatting in the current text style, first select text as follows:
  - To select one or more characters, click and drag the pointing device over the characters.
  - To select a word, double-click the word.
  - To select all the text in the cell, triple-click in the cell. (You can also right-click. Click Select All.)

6 On the toolbar, make format changes as follows:
  - To change the font of the selected text, select a font from the list.
  - To change the height of the selected text, enter a new value in the Height box.
  - To format text in a TrueType font with boldface or italics, or to create underlined text for any font, click the corresponding button on the toolbar. SHX fonts do *not* support boldface or italics.
  - To apply color to selected text, choose a color from the Color list. Click Other to display the Select Color dialog box.

7 Use the keyboard to move from cell to cell:
  - Press TAB to move to the next cell. In the last cell of the table, press TAB to add a new row.
  - Press SHIFT+TAB to move to the previous cell.
  - When the cursor is at the beginning or end of text in a cell, use the arrow keys to move the cursor to adjacent cells. You can also use CTRL+ an arrow key.
  - When text is highlighted in a cell, press an arrow key to remove selection and move the cursor to the beginning or end of the text in the cell.
  - Press ENTER to move down one cell.

8 To save your changes and exit, click OK on the toolbar or press CTRL+ENTER .

Command entry: DDEDIT
Shortcut menu: Select and right-click a cell. Click Edit Text.

To define or modify data formats

1. In a table, click the table cells where you want to redefine data and formatting.
2. On the Table toolbar, click Data Format.
3. Choose a data type, format, and other options for the selected table cells.
4. Enter data in the selected table cells. The data type and format you chose determines how the data is displayed.
5. Click OK.

Shortcut menu: Select and right-click a cell or range of cells. Click Data Format.

To change the properties of cells in a table

1. Click inside the table cell you want to change.

   Hold down SHIFT and click inside another cell to select those two cells and all the cells between them.

2. Use one of the following methods:
   - To change one or more properties, in the Properties palette, click the value you want to change and enter or select a new value.
   - To restore the default properties, right-click. Click Remove Property Overrides.

Toolbar: Properties 
Shortcut menu: Select and right-click a cell or range of cells. Click Properties.

To copy the properties of a cell to other cells

1. Click inside the table cell whose properties you want to copy.
2. (Optional) To view the current properties of the selected table cell, press CTRL+1 to open the Properties palette.

   All the properties of the cell are copied except the cell type: text or block.

3. On the Table toolbar, click Match Cell.

   The cursor changes to a paintbrush.

4 To copy the properties to another table cell in the drawing, click inside the cell.

5 Right-click or press ESC to stop copying properties.

Command entry: MATCHCELL
Shortcut menu: Select and right-click a cell or range of cells. Click Match Cell.

To change the lineweight, linetype, or color of the borders of table cells

1 Click inside the table cell you want to change.

   Hold down SHIFT and click inside another cell to select those two cells and all the cells between them.

2 On the Table toolbar, click Cell Borders.

3 In the Cell Border Properties dialog box, select a lineweight, linetype and color. To specify a double line border, select Double Line.

   Use BYBLOCK to set the border properties to match the settings in the table style that has been applied to the table.

4 Click one of the border type buttons to specify which borders of the cell to modify, or select a border in the preview image.

5 Click OK.

6 Move the cursor outside the Properties palette, and press ESC to remove selection, or select another cell.

Shortcut menu: Select and right-click a cell or range of cells. Click Borders.

To edit text in a table cell

1 Double-click inside the cell whose text you want to edit, or select the cell, right-click, and click Edit Cell.

   NOTE When a cell is selected, you can press F2 to edit the cell text.

2 Use the Text Formatting toolbar or the shortcut menu to make changes.

3 To save the changes and exit, click OK on the toolbar, press CTRL+ENTER, or click outside the cell.

4 To remove selection from the table, press ESC.

Command entry: DDEDIT
Shortcut menu: Select and right-click a cell. Click Edit Cell.

To insert a block in a table cell

1. On the Table toolbar, click Insert Block.
2. In the Insert dialog box, select a block from the list of blocks in the drawing, or click Browse to find a block in another drawing.
3. Specify the following properties 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.
   - **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.
4. Click OK.

   If the block has attributes attached, the Edit Attributes dialog box is displayed.

Shortcut menu: Select and right-click a cell. Click Insert ➤ Block.

To insert a field in a table cell

1. Double-click inside the table cell.
2. On the Table toolbar, click Insert Field, or press CTRL+F.
3. In the Field dialog box, select a category in the Field Category list to display the field names in that category.
4. Select a field.
5. Select the format or other options available for that field.
6. Click OK.

Command entry: FIELD
Shortcut menu: Select and right-click a cell. Click Insert ➤ Field.

Quick Reference

Commands

FIELD

Creates a multiline text object with a field that can be updated automatically as the field value changes.

FIND

Finds the text that you specify, and can optionally replace it with other text.

INSERT

Inserts a block or drawing into the current drawing.

LIST

Displays property data for selected objects.

MATCHCELL

Applies the properties of a selected table cell to other table cells.

MTEXT

Creates a multiline text object.

SPELL

Checks spelling in a drawing.

TABLE

Creates an empty table object.

TABLEDIT

Edits text in a table cell.

TABLEEXPORT

Exports data from a table object in CSV file format.

TABLESTYLE

Creates, modifies, or specifies table styles.

UPDATEFIELD

Manually updates fields in selected objects in the drawing.

System Variables

CTABLESTYLE

Sets the name of the current table style.

FIELDDISPLAY

Controls whether fields are displayed with a gray background.

FIELDEVAL

Controls how fields are updated.

Utilities

No entries

Command Modifiers

No entries

Use Formulas in Table Cells

Table cells can contain formulas that do calculations using the values in other table cells.

With a table cell selected, you can insert formulas from the Table toolbar as well as the shortcut menu. You can also open the In-Place Text Editor and enter a formula in a table cell manually.

Insert a Formula

In formulas, cells are referred to by their column letter and row number. For example, the cell at top left in the table is A1. Merged cells use the number of what would be the top-left cell. A range of cells is defined by the first and last cells, with a colon between them. For example, the range A5:C10 includes cells in rows 5 through 10 in columns A, B, and C.

A formula must start with an equal sign (=). The formulas for sum, average, and count ignore empty cells and cells that do not resolve to a numeric value. Other formulas display an error (#) if any cell in the arithmetic expression is empty or contains nonnumeric data.

Use the Cell option on the shortcut menu to select a cell in another table in the same drawing. When you have selected the cell, the In-Place Text Editor opens so you can enter the rest of the formula. You can also insert a formula using the Table toolbar.

Copy a Formula

When you copy a formula to another cell in the table, the range changes to reflect the new location. For example, if the formula in A10 sums A1 through A9, when you copy it to B10, the range of cells changes so that it sums B1 through B9.

If you don't want a cell address to change when you copy and paste the formula, add a dollar sign ($) to the column or row part of the address. For example, if you enter $A10, the column stays the same and the row changes. If you enter A10, both column and row stay the same.

Insert Data Automatically

You can automatically increment data in adjacent cells within a table by using the AutoFill grip. For example, a table with a date column can have the dates automatically entered by entering the first necessary date and dragging the AutoFill grip.

Numbers will fill automatically by increments of 1 if one cell is selected and dragged. Similarly, dates will resolve by increments of one day if only one cell is selected. If two cells are manually filled with dates one week apart, the remaining cells are incremented by one week.

See also:

- Use Fields in Text on page 1532

To sum the values in a range of table cells

1. Select the table cell where you want to place the formula by clicking inside it. The Table toolbar is displayed.
2. On the Table toolbar, click Insert Formula ➤ Sum.

   The following prompt is displayed:

   Select first corner of table cell range:
3. Click inside the first cell in the range.

   The following prompt is displayed:

   Select second corner of table cell range:
4. Click inside the last cell in the range.

   The In-Place Text Editor opens and displays the formula in the cell.
5. Edit the formula, if necessary.

6 To save your changes and exit the editor, use one of the following methods:
   - Click OK on the toolbar.
   - Click in the drawing outside the editor.
   - Press CTRL+ENTER.

   The cell displays the sum of the values in the range of cells. Empty cells and cells that do not resolve to a numeric value are ignored.

Shortcut menu: Select and right-click a cell or range of cells. Click Insert ➤ Formula ➤ Sum.

To average the values in a range of table cells

1. Select the table cell where you want to place the formula by clicking inside it. The Table toolbar is displayed.
2. On the Table toolbar, click Insert Formula ➤ Average.

   The following prompt is displayed:

   Select first corner of table cell range:
3. Click inside the first cell in the range.

   The following prompt is displayed:

   Select second corner of table cell range:
4. Click inside the last cell in the range.

   The In-Place Text Editor opens and displays the formula in the cell.
5. Edit the formula, if necessary.
6. To save your changes and exit the editor, use one of the following methods:
   - Click OK on the toolbar.
   - Click in the drawing outside the editor.
   - Press CTRL+ENTER.

   The cell displays the average of the values in the range of cells. Empty cells and cells that do not resolve to a numeric value are ignored.

Shortcut menu: Select and right-click a cell or range of cells. Click Insert ➤ Formula ➤ Average.

To count the cells in a table column or a table row

1. Select the table cell where you want to place the formula by clicking inside it. The Table toolbar is displayed.
2. On the Table toolbar, click Insert Formula ➤ Count

   The following prompt is displayed:

   Select first corner of table cell range:
3. Click inside the first cell in the range.

   The following prompt is displayed:

   Select second corner of table cell range:
4. Click inside the last cell in the range.

   The In-Place Text Editor opens and displays the formula in the cell.
5. Edit the formula, if necessary.
6. To save your changes and exit the editor, use one of the following methods:
   - Click OK on the toolbar.
   - Click in the drawing outside the editor.
   - Press CTRL+ENTER.

   The cell displays the total number of cells in the range.

Shortcut menu: Select and right-click a cell or range of cells. Click Insert ➤ Formula ➤ Count.

To use a cell from a different table in a formula

1. Select the table cell where you want to place the formula by clicking inside it. The Table toolbar is displayed.
2. On the Table toolbar, click Insert Formula ➤ Cell.

   The following prompt is displayed:

   Select table cell:
3. Click inside the cell in the other table.

   The In-Place Text Editor opens and displays the cell address.
4. Enter the rest of the formula.

5 To save your changes and exit the editor, use one of the following methods:
   - Click OK on the toolbar.
   - Click in the drawing outside the editor.
   - Press CTRL+ENTER.

   The cell displays the result of the calculation.

Shortcut menu: Select and right-click a cell or range of cells. Click Insert ➤ Formula ➤ Cell.

To manually enter a formula in a table cell

1 Double-click inside a table cell.

   The In-place Text Editor opens.

2 Enter a formula (a function or an arithmetic expression), as in the following examples:
   - **=sum(a1:a25,b1)**. Sums the values in the first 25 rows of column A and the first row in column B.
   - **=average(a100:d100)**. Calculates the average of the values in the first 4 columns in row 100.
   - **=count(a1:m500)**. Displays the total number of cells in column A through column M in rows 1 through 100.
   - **=(a6+d6)/e1**. Adds the values in A6 and D6 and divides the total by the value in E1.
     Use a colon to define a range of cells and a comma for individual cells. A formula must start with an equal sign (=) and can contain any of the following signs: plus (+), minus (-), times (*), divided by (/), exponent (^), and parentheses ().

3 To save your changes and exit the editor, use one of the following methods:
   - Click OK on the toolbar.
   - Click in the drawing outside the editor.
   - Press CTRL+ENTER.

   The cell displays the result of the calculation.

Shortcut menu: Select and right-click a cell or range of cells. Click Insert ➤ Formula ➤ Equation.

To turn off the display of column letters and row numbers for tables

1. At the command prompt, enter **tableindicator**.
2. At the Enter New Value prompt, enter **0**.

When TABLEINDICATOR is set to 1, the In-Place Text Editor displays column letters and row numbers when a table cell is selected.

Command entry: TABLEINDICATOR

To change the background color of column letters and row numbers for tables

1. Click a grid line to select a table.
2. Right-click. Click Table Indicator Color.
3. In the Select Color dialog box, select a color.
4. Click OK.

   The text color, size, and style and the line color are controlled by the settings for column heads in the current table style.

To insert a Formula field in a table cell

1. Click inside the table cell.
2. On the Table toolbar, click Insert Field.
3. In the Field dialog box, Field Category list, select Objects.
4. In Field Names, select Formula.
5. To enter a formula, use any of the following methods one or more times:
   - Click Average, Sum, or Count. The Field dialog box closes temporarily. To specify a range, click inside the first and the last cell. The result is appended to the formula.
   - Click Cell. The Field dialog box closes temporarily. Select a cell in a table in the drawing. The cell address is appended to the formula.
6. (Optional) Select a format and a decimal separator.
7. Click OK.

8 To save your changes and exit the editor, use one of the following methods:
   - Click OK on the toolbar.
   - Click in the drawing outside the editor.
   - Press CTRL+ENTER.

   The cell displays the result of the calculation.

Shortcut menu: Select and right-click a cell or range of cells. Click Insert ➤ Formula.

To automatically fill cells with incremented data

1 Double-click inside a table cell.

2 Enter a numeric value; for example, 1 or 01/01/2000.

3 Press the down arrow and enter the next desired numeric value.

4 On the Text Formatting toolbar, click OK.

   To change the format of the cell data, right-click the cell. Select Data Format.

5 Select the cell or cells from which you want to increment data from.

6 Click the grip in the lower right corner of the cell or cells.

   To change AutoFill options, right-click the AutoFill grip in the bottom right-hand corner of the selected cell range and select one of the following options:

   - **Fill Series**. Finds a pattern in the range of selected cells and fills the selected cells with the subsequent cell format and value. If only 1 cell is selected from which to increment data, then the data will increment at a rate of 1 unit.
   - **Fill Series Without Formatting**. Follows the above guidelines without cell formatting.
   - **Copy Cells**. Copies the format and values for the selected cells.
   - **Copy Cells Without Formatting**. Follows the above guidelines without cell formatting.
   - **Fill Formatting Only**. Fills only cell formatting for the selected cells. Cell values are ignored.

7 Drag the grip through the cells you would like to automatically increment. A preview of the value for each cell will display to the right of the selected grip.

Quick Reference

Commands

FIELD

Creates a multiline text object with a field that can be updated automatically as the field value changes.

MTEXT

Creates a multiline text object.

TABLE

Creates an empty table object.

TABLEEXPORT

Exports data from a table object in CSV file format.

TABLESTYLE

Creates, modifies, or specifies table styles.

UPDATEFIELD

Manually updates fields in selected objects in the drawing.

System Variables

CTABLESTYLE

Sets the name of the current table style.

FIELDDISPLAY

Controls whether fields are displayed with a gray background.

FIELDEVAL

Controls how fields are updated.

TABLEINDICATOR

Controls the display of row numbers and column letters when the In-Place Text Editor is open for editing a table cell.

Utilities

No entries

Command Modifiers

No entries

Dimensions and Tolerances 27

You can add measurements to your drawing with several dimensioning commands. Use dimension styles to format dimensions quickly and maintain industry or project dimensioning standards.

Understand Basic Concepts of Dimensioning

You can create several types of dimensions, and you can control their appearance by setting up dimension styles or by editing individual dimensions.

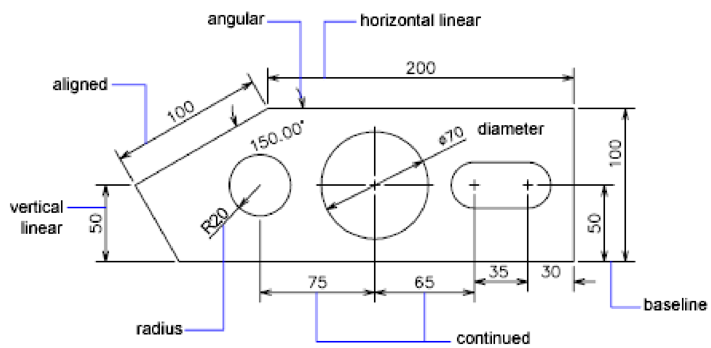
Overview of Dimensioning

Dimensioning is the process of adding measurement annotation to a drawing.

You can create dimensions for a variety of object types in many orientations. The basic types of dimensioning are

- Linear
- Radial (radius, diameter and jogged)
- Angular
- Ordinate
- Arc Length

Linear dimensions can be horizontal, vertical, aligned, rotated, baseline, or continued (chained). Some examples are shown in the illustration.



NOTE To simplify drawing organization and dimension scaling, it is recommended that you create dimensions on layouts rather than in model space.

To create a dimension

1. Create a layer designated for dimensions and make it the current layer.
2. Near the bottom-left corner of the application window, click a layout tab.
3. Click Dimension menu. Click a dimension option.
4. Follow the command prompts.

Command entry: DIMLINEAR, DIMRADIUS, DIMANGULAR

Quick Reference

Commands

DIMANGULAR

Creates an angular dimension.

DIMARC

Creates an arc length dimension.

DIMBREAK

Breaks or restores dimension and extension lines where they cross other objects.

DIMDIAMETER

Creates a diameter dimension for a circle or an arc.

DIMEDIT

Edits dimension text and extension lines.

DIMBREAK

Adds or removes inspection information for a selected dimension.

DIMJOGGED

Creates jogged dimensions for circles and arcs.

DIMBREAK

Adds or removes a jog line on a linear or aligned dimension.

DIMLINEAR

Creates a linear dimension.

DIMORDINATE

Creates ordinate dimensions.

DIMRADIUS

Creates a radius dimension for a circle or an arc.

DIMREASSOCIATE

Associates or reassociates selected dimensions to objects or points on objects.

DIMBREAK

Adjusts the spacing between linear dimensions or angular dimensions.

DIMSTYLE

Creates and modifies dimension styles.

DIMTEDIT

Moves and rotates dimension text and relocates the dimension line.

PROPERTIES

Controls properties of existing objects.

QDIM

Quickly creates a series of dimensions from selected objects.

System Variables

DIMASSOC

Controls the associativity of dimension objects and whether dimensions are exploded.

Utilities

No entries

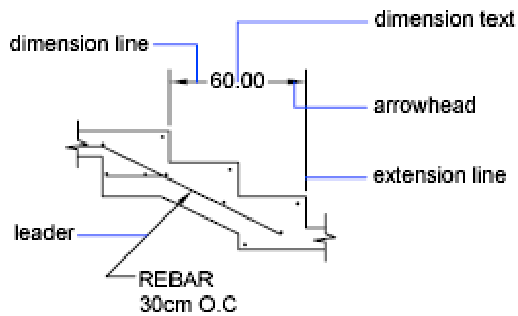
Command Modifiers

No entries

Parts of a Dimension

Here is a list of the parts of a dimension along with their descriptions.

Dimensions have several distinct elements: dimension text, dimension lines, arrowheads, and extension lines.



Dimension text is a text string that usually indicates the measurement value. The text can also include prefixes, suffixes, and tolerances.

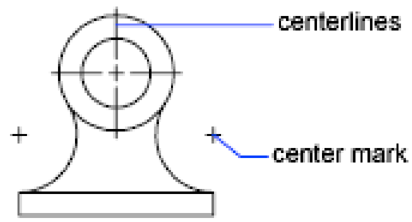
A *dimension line* indicates the direction and extent of a dimension. For angular dimensions, the dimension line is an arc.

Arrowheads, also called symbols of termination, are displayed at each end of the dimension line. You can specify different sizes and shapes for arrowheads or tick marks.

Extension lines, also called projection lines or witness lines, extend from the feature to the dimension line.

A *center mark* is a small cross that marks the center of a circle or arc.

Centerlines are broken lines that mark the center of a circle or arc.



Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Associative Dimensions

Dimensions can be associative, nonassociative, or exploded. Associative dimensions adjust to changes in the geometric objects that they measure.

Dimension associativity defines the relationship between geometric objects and the dimensions that give their distance and angles. There are three types of associativity between geometric objects and dimensions.

- **Associative dimensions.** Automatically adjust their locations, orientations, and measurement values when the geometric objects associated with them are modified. Dimensions in a layout may be associated to objects in model space. The DIMASSOC system variable is set to 2.

- **Nonassociative dimensions.** Selected and modified with the geometry they measure. Nonassociative dimensions do not change when the geometric objects they measure are modified. The dimension variable DIMASSOC is set to 1.
- **Exploded dimensions.** Contain a collection of separate objects rather than a single dimension object. The DIMASSOC system variable is set to 0.

You can determine whether a dimension is associative or nonassociative by selecting the dimension and doing one of the following:

- Use the Properties palette to display the properties of the dimension.
- Use the LIST command to display the properties of the dimension.

You can also use the Quick Select dialog box to filter the selection of associative or nonassociative dimensions. A dimension is considered associative even if only one end of the dimension is associated with a geometric object. The DIMREASSOCIATE command displays the associative and nonassociative elements of a dimension.

Special Situations and Limitations

You may need to use DIMREGEN to update associative dimensions after panning or zooming with a wheel mouse, after opening a drawing that was modified with an earlier release, or after opening a drawing with external references that have been modified.

Although associative dimensions support most object types that you would expect to dimension, they do not support the following:

- Hatches
- Multiline objects
- 2D solids
- Objects with nonzero thickness
- Images
- DWF underlays

When selecting objects to dimension, make sure that the objects that you select don't include a directly overlapping object that does not support associative dimensioning such as a 2D solid.

Associativity is *not* maintained between a dimension and a block reference if the block is redefined.

Associativity is not maintained between a dimension and a 3D solid if the shape of the 3D solid is modified.

Dimensions created with QDIM are not associative but may be associated individually with DIMREASSOCIATE.

NOTE In releases prior to AutoCAD 2002, the definitions of associative and nonassociative dimensions were different and were controlled by the DIMASO system variable. The behavior of dimensions is now controlled by the DIMASSOC system variable.

For information about working with associative dimensions in combination with previous releases, see Save Drawings to Previous Drawing File Formats.

See also:

- Change Dimension Associativity on page 1733

To change the dimension associativity default

1. Click Tools menu ➤ Options.
2. In the Options dialog box, User Preferences tab, under Associative Dimensioning, select or clear Make New Dimensions Associative.
3. Do either or both of the following:
   - Click Apply to record the current Options settings in the system registry.
   - Click OK to record the current Options settings in the system registry and close the Options dialog box.

All subsequently created dimensions in the drawing use the new setting. Unlike most other option settings, dimension associativity is saved in the drawing file rather than in the system registry.

Command entry: OPTIONS

Quick Reference

Commands

DIMDISASSOCIATE

Removes associativity from selected dimensions.

DIMREASSOCIATE

Associates or reassociates selected dimensions to objects or points on objects.

DIMREGEN

Updates the locations of all associative dimensions.

EXPLODE

Breaks a compound object into its component objects.

LIST

Displays property data for selected objects.

OPTIONS

Customizes the program settings.

System Variables

DIMASSOC

Controls the associativity of dimension objects and whether dimensions are exploded.

Utilities

No entries

Command Modifiers

No entries

Use Dimension Styles

You can control the appearance of dimensions by changing settings. For convenience and to help maintain dimensioning standards, you can store these settings in dimension styles.

Overview of Dimension Styles

A dimension style is a named collection of dimension settings that controls the appearance of dimensions, such as arrowhead style, text location, and lateral tolerances.

You create dimension styles to specify the format of dimensions quickly, and to ensure that dimensions conform to industry or project standards.

- When you create a dimension, it uses the settings of the current dimension style
- If you change a setting in a dimension style, all dimensions in a drawing that use the style update automatically
- You can create *dimension substyles* that, for specified types of dimensions, deviate from the current dimension style
- If necessary, you can override a dimension style temporarily

To set the current dimension style

- On the Styles toolbar, in the Dimension Styles control, click the arrow and select a dimension style from the list.

Command entry: DIMSTYLE

To create a dimension substyle

1 Click Home tab ➤ Annotation panel ➤ Dimension Style.

2 In the Dimension Style Manager, select the style from which you want to create a substyle. Click New.

3 In the Create New Dimension Style dialog box, select the type of dimension that will apply to the substyle from the Use For list. Click Continue.

4 In the New Dimension Style dialog box, select the appropriate tab and make changes to define the dimension substyle.

5 Click OK.

6 Click Close to exit the Dimension Style Manager.

Command entry: DIMSTYLE

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Compare Dimension Styles and Variables

You can view all the settings in a dimension style. Dimension styles used in externally referenced drawings are differentiated from those defined in your current drawing.

You can list the dimension styles in the current drawing. You can also list all dimensioning system variables and their current status or only the variables affected by a dimension style.

When you list the current status of all dimensioning system variables, any running overrides that apply to the current dimension style are listed. You can also list the differences between a named dimension style and the current dimension style.

Use Externally Referenced Dimension Styles

The program displays externally referenced dimension style names using the same syntax as for other externally dependent named objects. When you view externally referenced dimension styles using the Dimension Style Manager, the name of the xref displays in the Styles list as *Xref:drawing name* with each xref style appearing below the drawing name. For example, if the drawing file *baseplat.dwg* has a dimension style called FRACTIONAL-1, and you attach *baseplat.dwg* as an xref to a new drawing, then the xref dimension style is displayed in the Styles list of the Dimension Style Manager as *Xref:baseplat.dwg*, and FRACTIONAL-1 appears under the drawing name.

Externally referenced dimension styles can be examined, but they cannot be modified or made current. You can use an externally referenced dimension style as a template for creating a new dimension style in your current drawing.

To list all dimension settings for the current dimension style

1. Click Home tab ➤ Annotation panel ➤ Dimension Style.
2. In the Dimension Style Manager, select the style from the Styles list.
3. Click Compare.

   The dimensioning system variables, their current settings, and a brief description are listed. Overrides are included.

Toolbar: Dimension or Styles
Command entry: DIMSTYLE

To list settings for an existing dimension style

1. Click Home tab ➤ Annotation panel ➤ Dimension Style.
2. In the Dimension Style Manager, enter a dimension style name, or select a dimension whose dimension style you want to examine.
3. Click Compare.

   Affected variables, their settings, and a brief description of each are listed. Overrides are not included.

Toolbar: Dimension or Styles
Command entry: DIMSTYLE

To list dimension styles in the current drawing

1. Click Home tab ➤ Annotation panel ➤ Dimension Style.

2 In the Dimension Style Manager, under List, select All Styles or Style in Use.

Toolbar: Dimension or Styles
Command entry: DIMSTYLE

To compare dimension styles

1 Click Home tab ➤ Annotation panel ➤ Dimension Style.

2 In the Dimension Style Manager, select the style to compare from the Styles list.

3 Click Compare.

The dimension style is compared to the current dimension style.

Toolbar: Dimension or Styles
Command entry: DIMSTYLE

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Control Dimension Geometry

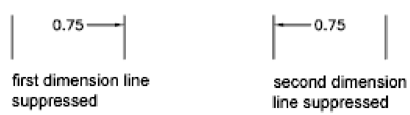
You can control the appearance of dimension lines, extension lines, arrowheads, and center marks.

Control Dimension Lines

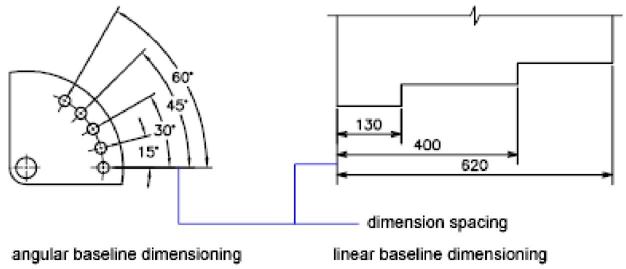
You can control dimension line properties including color, lineweight, and spacing.

You can control several aspects of a dimension line. You can

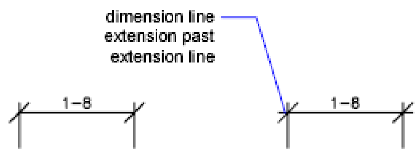
- Specify color and lineweight for visual effect and plotting
- Suppress the dimension line or, if the dimension line is broken by text, one or both halves



- Control the spacing between successive dimension lines in baseline dimensions



- Control the distance by which the dimension line extends beyond the extension lines for architectural tick (oblique stroke) arrowheads



To modify the display of dimension lines

1. Click Home tab ➤ Annotation panel ➤ Dimension Style. 
2. In the Dimension Style Manager, select the style you want to change. Click Modify.
3. In the Modify Dimension Style dialog box, Lines tab, change the settings under Dimension Lines as needed.
4. Click OK.
5. Click Close to exit the Dimension Style Manager.

Toolbar: Dimension or Styles 
Command entry: DIMSTYLE

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

System Variables

DIMCLRD

Assigns colors to dimension lines, arrowheads, and dimension leader lines.

DIMDLE

Sets the distance the dimension line extends beyond the extension line when oblique strokes are drawn instead of arrowheads.

DIMDLI

Controls the spacing of the dimension lines in baseline dimensions.

DIMGAP

Sets the distance around the dimension text when the dimension line breaks to accommodate dimension text.

DIMLTYPE

Sets the linetype of the dimension line.

DIMLWD

Assigns lineweight to dimension lines.

DIMSD1

Controls suppression of the first dimension line and arrowhead.

DIMSD2

Controls suppression of the second dimension line and arrowhead.

DIMSOXD

Suppresses arrowheads if not enough space is available inside the extension lines.

DIMTOFL

Controls whether a dimension line is drawn between the extension lines even when the text is placed outside.

Utilities

No entries

Command Modifiers

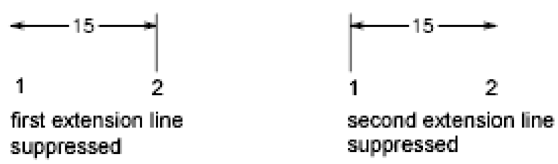
No entries

Control Extension Lines

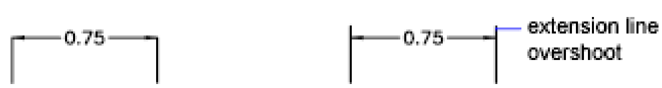
You can control extension line properties including color, lineweight, overshoot, and offset length.

You can control several aspects of the extension lines. You can

- Specify color and lineweight for visual effect and plotting
- Suppress one or both extension lines if they are unnecessary, or if there is not enough space

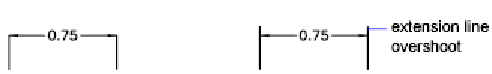


- Specify how far beyond from the dimension line the extension line extends (overshoot)



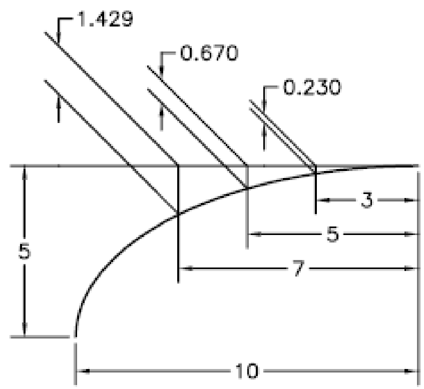
- Control the extension origin offset, the distance between the extension line origin, and the start of the extension line

- Specify a fixed length for extension lines, as measured from the dimension line toward the extension line origin



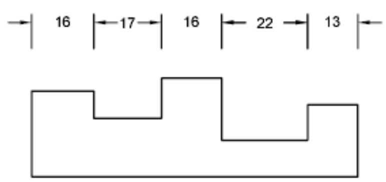
- Specify a noncontinuous linetype, typically used for centerlines

- Modify the angle of the extension lines of a selected dimension to make them oblique



Fixed-Length Extension Lines

With the Dimension Style Manager, on the Lines tab, you can specify a dimension style that sets the total length of extension lines starting from the dimension line toward the dimension origin point.



fixed-length extension lines

The extension-line offset distance from the origin will never be less than the value specified by the DIMEXO system variable.

See also:

- Create Dimensions with Oblique Extension Lines on page 1686

To modify the display of extension lines

1. Click Home tab ➤ Annotation panel ➤ Dimension Style.
2. In the Dimension Style Manager, select the style you want to change. Click Modify.
3. In the Modify Dimension Style dialog box, Lines tab, under Extension Lines, change the settings as needed.
4. Click OK.
5. Click Close to exit the Dimension Style Manager.

Toolbar: Dimension or Styles
Command entry: DIMSTYLE

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

System Variables

DIMCLRE

Assigns colors to dimension extension lines.

DIMDLE

Sets the distance the dimension line extends beyond the extension line when oblique strokes are drawn instead of arrowheads.

DIMEXE

Specifies how far to extend the extension line beyond the dimension line.

DIMEXO

Specifies how far extension lines are offset from origin points.

DIMFXL

Sets the total length of the extension lines starting from the dimension line toward the dimension origin.

DIMFXLON

Controls whether extension lines are set to a fixed length.

DIMLTEX1

Sets the linetype of the first extension line.

DIMLTEX2

Sets the linetype of the second extension line.

DIMLWE

Assigns lineweight to extension lines.

DIMSE1

Suppresses display of the first extension line.

DIMSE2

Suppresses display of the second extension line.

Utilities

No entries

Command Modifiers

No entries

Control Dimension Arrowheads

You can control the arrowhead symbols in dimensions and leaders including their type, size, and visibility.

You can choose from many standard types of arrowheads, or you can create your own arrowheads. Additionally, you can

- Suppress the display of arrowheads, or use one arrowhead only
- Apply a different type of arrowhead to each end of a dimension line
- Control the size of arrowheads

- Flip the direction of an arrowhead using the dimension shortcut menu

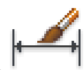
---

NOTE Flipped arrowheads maintain their appearance in versions later than AutoCAD 2002. However, if you edit a drawing with flipped arrowheads in a release earlier than AutoCAD 2006, the arrowhead directions will revert to their original orientations.

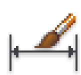
See also:

- Customize Arrowheads on page 1643

To choose an arrowhead

1. Click Home tab ➤ Annotation panel ➤ Dimension Style. 
2. In the Dimension Style Manager, select the style you want to change. Click Modify.
3. In the Modify Dimension Style dialog box, Symbols and Arrows tab, under Arrowheads, select the arrowhead type for the first end of the dimension line.

   The second arrowhead is automatically set to the same type.
4. To set the second end of the dimension line to a different arrowhead type, select an arrowhead type from the Second list.
5. In the Size box, enter a size for the arrowhead.
6. Click OK.
7. Click Close to exit the Dimension Style Manager.

Toolbar: Dimension or Styles 
Command entry: DIMSTYLE

To flip the direction of an arrowhead

1. At the command prompt, select a single dimension object near the arrowhead that you want to flip.
2. Right-click. Click Flip Arrow.

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

System Variables

DIMCLRD

Assigns colors to dimension lines, arrowheads, and dimension leader lines.

DIMDLE

Sets the distance the dimension line extends beyond the extension line when oblique strokes are drawn instead of arrowheads.

DIMSD1

Controls suppression of the first dimension line and arrowhead.

DIMSD2

Controls suppression of the second dimension line and arrowhead.

Utilities

No entries

Command Modifiers

No entries

Customize Arrowheads

You can create your own custom arrowheads.

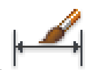
Arrowheads are stored as block definitions. To use your own arrowhead, provide the name of an existing block definition. For information about creating blocks, see Create Blocks Within a Drawing on page 855.

NOTE Annotative blocks cannot be used as custom arrowheads for dimensions or leaders.

Arrowhead sizing relies on the overall dimension scale factor. When you create a dimension, the block is inserted where the arrowheads would normally go. The object's *X* and *Y* scale factors are set to *arrowhead size overall scale*. The dimension line is trimmed by *text gap x overall scale* units at each end. To trim the dimension line, the rightmost block is inserted with a zero rotation angle for horizontal dimensioning. The leftmost block is rotated 180 degrees about its insertion point.

If you use paper-space scaling, the scale factor is computed before applying it to the arrowhead size value.

To use your own arrowhead symbol

1 Click Home tab ➤ Annotation panel ➤ Dimension Style. 

2 In the Dimension Style Manager, select the style you want to change. Click Modify.

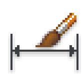
3 In the Dimension Style Manager, Symbols and Arrows tab, under Arrowheads, select User Arrow from the First arrowhead list.

4 In the Select Custom Arrow Block dialog box, enter the name of your block. Click OK.

5 To choose a different custom arrowhead for the second arrowhead, repeat steps 3 and 4, choosing User Arrow from the Second arrowhead list (optional).

6 Click OK.

7 Click Close to exit the Dimension Style Manager.

Toolbar: Dimension or Styles 
Command entry: DIMSTYLE

Quick Reference

Commands

BLOCK

Creates a block definition from selected objects.

DIMSTYLE

Creates and modifies dimension styles.

WBLOCK

Writes objects or a block to a new drawing file.

System Variables

DIMASZ

Controls the size of dimension line and leader line arrowheads.

DIMBLK

Sets the arrowhead block displayed at the ends of dimension lines or leader lines.

DIMBLK1

Sets the arrowhead for the first end of the dimension line when DIMSAH is on.

DIMBLK2

Sets the arrowhead for the second end of the dimension line when DIMSAH is on.

DIMCLRD

Assigns colors to dimension lines, arrowheads, and dimension leader lines.

DIMDLE

Sets the distance the dimension line extends beyond the extension line when oblique strokes are drawn instead of arrowheads.

DIMSAH

Controls the display of dimension line arrowhead blocks.

DIMTSZ

Specifies the size of oblique strokes drawn instead of arrowheads for linear, radius, and diameter dimensioning.

Utilities

No entries

Command Modifiers

No entries

Control Dimension Text

You can control the placement of dimension text, arrowheads, and leader lines relative to the dimension and extension lines.

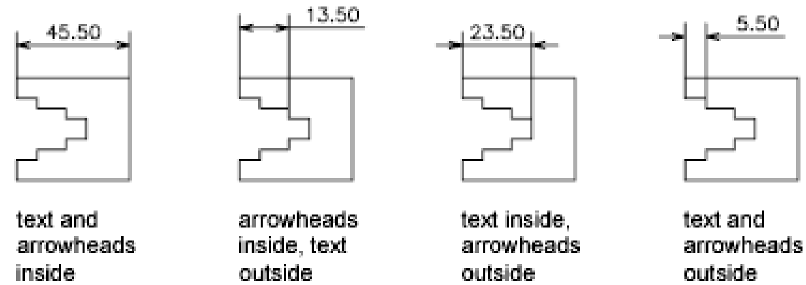
Fit Dimension Text Within Extension Lines

Dimension text and arrowheads usually appear between the extension lines when there is enough space. You can specify how these elements are placed when space is limited.

Many factors, such as the size of extension line spacing and arrowhead size, influence how dimension text and arrowheads fit within the extension lines. In general, the best fit, given the available space, is applied. If possible, both text and arrowheads are accommodated between the extension lines, no matter what fit option you choose.

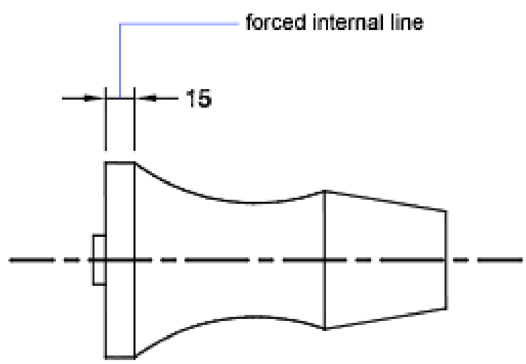
When creating new dimensions, you can choose to place text by entering a coordinate or using the pointing device; this is known as user-defined text placement. Alternatively, the program can compute the text position for you. The options for automatic fitting of text and arrowheads are listed in the Dimension Style Manager, Fit tab. For example, you can specify that text and arrowheads be kept together. In this case, if there is not room for both between the extension lines, they are both placed outside. You can specify that if there is room for only text or arrowheads, then either text only or arrowheads only are placed between the extension lines.

The following illustrations show how the program applies a "best fit" for arrowheads and text.



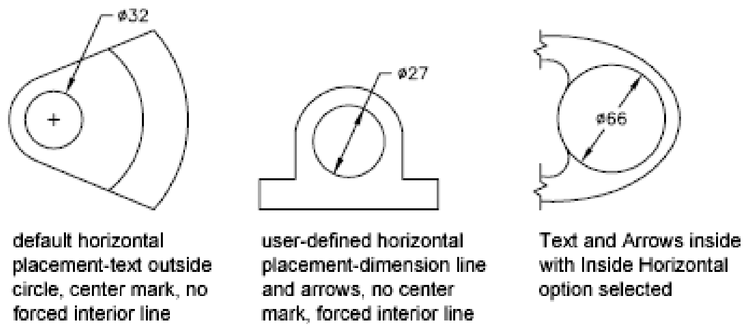
If there is no room for text between the extension lines, you can have a leader line created automatically. This is useful in cases where text outside the extension lines would interfere with other geometry, for example, in continued dimensions. Whether text is drawn to the right or the left of the leader is controlled by the horizontal justification setting on the Text tab of the Modify/New Dimension Style dialog box. Also, you can fit text and arrowheads by changing their size.

Even if the arrowheads are outside the extension lines, you can have a line drawn between the extension lines. This is called forcing an internal line and is illustrated as follows.



Fit Diameter Dimension Text

You can draw several different diameter dimensions depending on text placement, horizontal settings on the Text tab, and whether you select the Draw Dim Line Between Ext Lines option on the Fit tab.



To place text within extension lines

1 Click Home tab ➤ Annotation panel ➤ Dimension Style.

2 In the Dimension Style Manager, select the style you want to change. Click Modify.

3 In the Modify Dimension Style dialog box, Fit tab, under Fit Options, select an option.

4 Click OK.

5 Click Close to exit the Dimension Style Manager.
If there is enough room, text is fit between extension lines.

Toolbar: Dimension or Styles
Command entry: DIMSTYLE

To force an internal line and choose a fit option

1 Click Home tab ➤ Annotation panel ➤ Dimension Style.

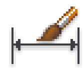
2 In the Dimension Style Manager, select the style you want to change. Click Modify.

3 In the Modify Dimension Style dialog box, Fit tab, Under Fine Tuning, select Always Draw Dim Line Between Ext Lines.

4 Under Fit Options, select an option.

5 Click OK.

6 Click Close to exit the Dimension Style Manager.

Toolbar: Dimension or Styles 
Command entry: DIMSTYLE

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

System Variables

DIMATFIT

Determines how dimension text and arrows are arranged when space is not sufficient to place both within the extension lines.

DIMJUST

Controls the horizontal positioning of dimension text.

DIMLWD

Assigns lineweight to dimension lines.

DIMTAD

Controls the vertical position of text in relation to the dimension line.

DIMTXTDIRECTION

Specifies the reading direction of the dimension text.

DIMTIH

Controls the position of dimension text inside the extension lines for all dimension types except Ordinate.

DIMTIX

Draws text between extension lines.

DIMTOFL

Controls whether a dimension line is drawn between the extension lines even when the text is placed outside.

DIMTOH

Controls the position of dimension text outside the extension lines.

DIMTVP

Controls the vertical position of dimension text above or below the dimension line.

DIMUPT

Controls options for user-positioned text.

Utilities

No entries

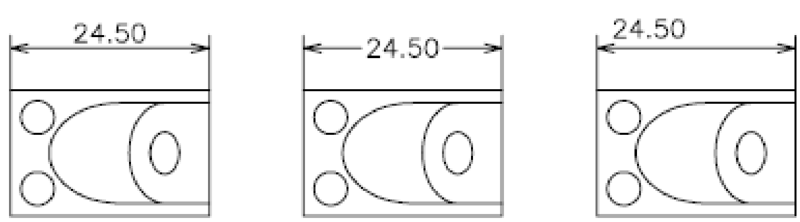
Command Modifiers

No entries

Control the Location of Dimension Text

You can locate dimension text manually and specify its alignment and orientation.

The program comes with several justification settings that facilitate compliance with international standards, or you can choose your own location for the text.



text centered above the dimension line

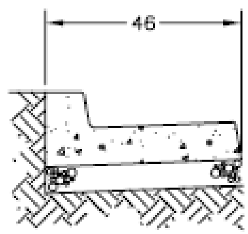
text centered horizontally and vertically

text left-justified above the dimension line

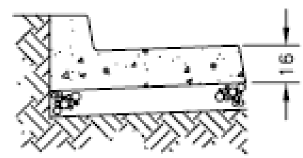
Many of the settings are interdependent. Example images in the Dimension Style Manager are updated dynamically to illustrate how text appears as you change the settings.

Align Dimension Text

Whether text is inside or outside the extension lines, you can choose whether it is aligned with the dimension line or remains horizontal. The following examples show two combinations of these options.



text inside extension lines oriented horizontally

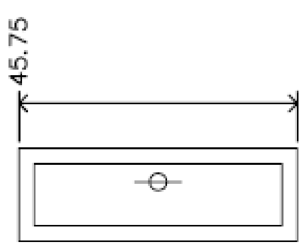


text outside extension lines aligned with dimension line

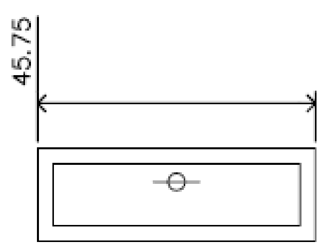
The default alignment is horizontal dimension text, even for vertical dimensions.

Position Dimension Text Horizontally

The position of the text along the dimension line in relation to the extension lines is referred to as text placement. To place text yourself when you create a dimension, use the Place Text Manually When Dimensioning option on the Fit tab of the Modify/New Dimension Style dialog box. Use the text placement options to automatically place text at the center of the dimension line, at either extension line, or over either extension line.

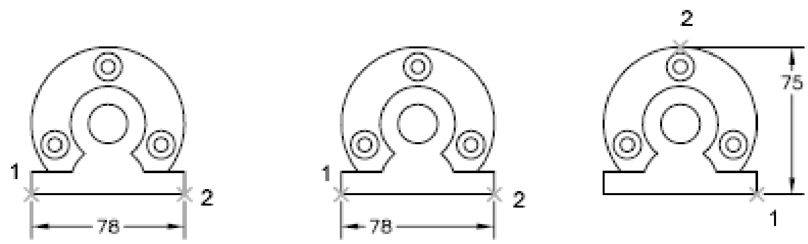


horizontal justification over extension line one, vertical justification of text, Centered over extension line



horizontal justification over extension line one, vertical justification of text, Above extension line

First and *second* extension lines are defined by the order in which you specified the extension line origins when you created the dimension. For angular dimensions, the second extension line is counterclockwise from the first. In the following illustrations, 1 is the first extension line origin and 2 the second.



text horizontal and centered along dimension line

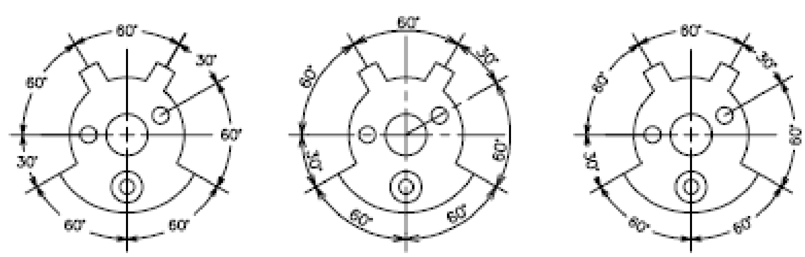
text horizontal at first extension line

text horizontal at second extension line

If you place text manually, you can place the dimension text anywhere along the dimension line, inside or outside the extension lines, as you create the dimension. This option provides flexibility and is especially useful when space is limited. However, the horizontal alignment options provide better accuracy and consistency between dimensions.

Position Dimension Text Vertically

The position of the text relative to the dimension line is referred to as vertical text placement. Text can be placed above or below or centered within the dimension line. In the ANSI standards, centered text usually splits the dimension line. In the ISO standards, it is usually above or outside the dimension line. For example, ISO standards permit angular dimension text to appear in any of the ways shown.



ANSI standard text centered within the dimension line

ISO standard text above the dimension line

text aligned with and centered within the dimension line

Other settings, such as Text Alignment, affect the vertical alignment of text. For example, if Horizontal Alignment is selected, text inside the extension lines and centered within the dimension line is horizontal, as shown in the leftmost illustration above. The text is horizontal even if the dimension line is not itself horizontal.

To align text with the dimension line

1 Click Home tab ➤ Annotation panel ➤ Dimension Style.

2 In the Dimension Style Manager, select the style you want to change. Click Modify.

3 In the Modify Dimension Style dialog box, Text tab, under Text Alignment, select Aligned with Dimension Line.

4 Click OK.

5 Click Close to exit the Dimension Style Manager.

Toolbar: Dimension or Styles
Command entry: DIMSTYLE

To place text at the second extension line

1 Click Home tab ➤ Annotation panel ➤ Dimension Style.

2 In the Dimension Style Manager, select the style you want to change. Click Modify.

3 In the Modify Dimension Style dialog box, Text tab, under Text Placement, select Over Ext Line 2 from the Horizontal list box.

   The example area reflects your selection.

4 Click OK.

5 Click Close to exit the Dimension Style Manager.

Toolbar: Dimension or Styles

Command entry: DIMSTYLE

To place dimension text manually

1 Click Home tab ➤ Annotation panel ➤ Dimension Style.

2 In the Dimension Style Manager, select the style you want to change. Click Modify.

3 In the Modify Dimension Style dialog box, Fit tab, under Fine Tuning, select Place Text Manually When Dimensioning.

4 Click OK.

5 Click Close to exit the Dimension Style Manager.

As you create dimensions, you can move the text along the dimension line. Use the pointing device or enter coordinates to specify the dimension line and text locations.

Toolbar: Dimension or Styles
Command entry: DIMSTYLE

To place text above the dimension line

1 Click Home tab ➤ Annotation panel ➤ Dimension Style.

2 In the Dimension Style Manager, select the style you want to change. Click Modify.

3 In the Modify Dimension Style dialog box, Text tab, under Text Placement, select Above from the Vertical list box.

The example area reflects your selection.

4 Click OK.

5 Click Close to exit the Dimension Style Manager.

Toolbar: Dimension or Styles
Command entry: DIMSTYLE

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

DIMTEDIT

Moves and rotates dimension text and relocates the dimension line.

System Variables

DIMJUST

Controls the horizontal positioning of dimension text.

DIMTAD

Controls the vertical position of text in relation to the dimension line.

DIMTXTDIRECTION

Specifies the reading direction of the dimension text.

DIMTIH

Controls the position of dimension text inside the extension lines for all dimension types except Ordinate.

DIMTOH

Controls the position of dimension text outside the extension lines.

DIMTVP

Controls the vertical position of dimension text above or below the dimension line.

DIMUPT

Controls options for user-positioned text.

Utilities

No entries

Command Modifiers

No entries

Control the Appearance of Dimension Text

You can include prefixes, suffixes, and user-supplied text in dimensions. You can also control the text style and formatting used in dimension text.

The program supports a mixture of user-supplied text, prefixes and suffixes supplied by the dimension style, and generated measurements. For example, you could add a diameter symbol as a prefix to a measurement or add the abbreviation for a unit, such as mm, as a suffix. Text in this context refers to all dimension text, prefixes and suffixes, primary and alternate units, and lateral tolerances. Geometric tolerances are controlled independently.

Dimension text is treated as a single string of text, which you create and format using your text editor.

Control the Text Style in Dimensions

The appearance of dimension text is governed by the text style selected in the Dimension Style Manager, Text tab. You can choose a text style while creating a dimension style and specify a text color and a height independent of the current text style's height setting. You can also specify the gap between base dimension text and the box that surrounds it.

The text styles used for dimensions are the same text styles used by all text created in your drawing.

For more information, see Work with Text Styles on page 1544.

Supply User Text to Dimensions

In addition to the prefixes and suffixes specified for primary and alternate units, you can supply your own text as you create a dimension. Because the prefix, suffix, and user-supplied text form a single text string, you can represent tolerance stacks and apply changes to font, text size, and other characteristics using the text editor.

To add user text above and below the dimension line, use the separator symbol \X. Text that precedes this symbol is aligned with and above the dimension line. Text that follows the \X symbol is aligned with and below the dimension line. The space between the dimension line and the text is determined by the value you enter under Gap in the Annotation dialog box.

Example: User Text in Dimensions

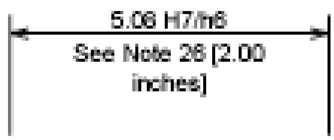
In this example, the primary dimension measurement is 5.08, and the alternate dimension measurement is 2.00. The primary units have the suffix *H7/h6*, and the alternate units have the suffix *inches*.

At the text prompt, while creating the dimension, you enter the following format string:

<> H7/h6\XSee Note 26\P[ ]

The angle brackets represent the primary units, and the square brackets represent the alternate units. The \X separates text above the dimension line from text below the dimension line. The \P is a paragraph break.

The resulting text appears as follows:



To control the text style in dimensions

1. Click Home tab ➤ Annotation panel ➤ Dimension Style. 
2. In the Dimension Style Manager, select the style you want to change. Click Modify.
3. In the Modify Dimension Style dialog box, Text tab, under Text Appearance, select a text style.
4. If the current text style does not have a fixed height, enter the height of dimension text in the Text Height box.
5. Under Tolerances, enter a height for tolerance values in the Scaling for Height box.
6. In the Offset from Dim Line box, enter a value for the gap around base dimension text.
7. Select a color from the Text Color box.

8 Click OK

9 Click Close to exit the Dimension Style Manager.

Toolbar: Dimension or Styles
Command entry: DIMSTYLE

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

System Variables

DIMCLRT

Assigns colors to dimension text.

DIMGAP

Sets the distance around the dimension text when the dimension line breaks to accommodate dimension text.

DIMTFAC

Specifies a scale factor for the text height of fractions and tolerance values relative to the dimension text height, as set by DIMTXT.

DIMTFILL

Controls the background of dimension text.

DIMTFILLCLR

Sets the color for the text background in dimensions.

DIMTXSTY

Specifies the text style of the dimension.

DIMTXT

Specifies the height of dimension text, unless the current text style has a fixed height.

DIMTXTDIRECTION

Specifies the reading direction of the dimension text.

Utilities

No entries

Command Modifiers

No entries

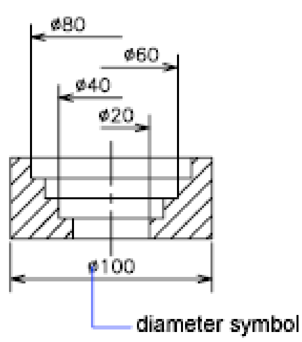
Control Dimension Values

The numeric values displayed in dimensions can appear in several formats. You can also control how numeric distances are represented.

Control the Display of Dimension Units

The numeric values of dimensions can be displayed as a single measurement or in two measurement systems. In either case, you can control details of how the numeric values are presented.

The settings for primary units control the display of the dimension values, including the unit format, the numeric precision, and the decimal separator style. For example, you can enter the diameter symbol as a prefix, as shown in the illustration. Any prefix you specify replaces the prefixes normally used for diameter and radius dimensions (Diameter (unicode 2205 and R, respectively).



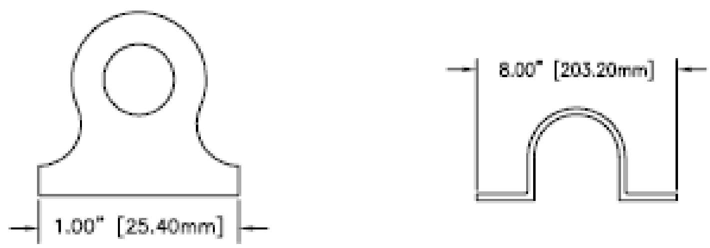
These settings are available on the Primary Units tab of the Dimension Style Manager.

Control the Display of Alternate Units

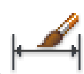
You can create dimensions in two systems of measurement simultaneously. A common use of this feature is to add feet and inches dimensions to drawings created using metric units. The alternate units appear in square brackets ([ ]) in the dimension text. Alternate units cannot be applied to angular dimensions.

If alternate-units dimensioning is on when you edit a linear dimension, the measurement is multiplied by an alternate scale value that you specify. This value represents the number of alternate units per current unit of measurement. The default value for imperial units is 25.4, which is the number of millimeters per inch. The default value for metric units is about 0.0394, which is the number of inches per millimeter. The number of decimal places is specified by the precision value for alternate units.

For example, for imperial units, if the alternate scale setting is the default value, 25.4, and the alternate precision is 0.00, the dimension might look like the following figure.



To add and format primary units

1 Click Home tab ➤ Annotation panel ➤ Dimension Style. 

2 In the Dimension Style Manager, select the style you want to change. Click Modify.

3 In the Modify Dimension Style dialog box, Primary Units tab, under Linear or Angular Dimensions, select a unit format and precision value for the primary units.

4 Under Linear Dimensions, enter any prefix and suffix for the displayed dimension.

5 Click OK.

6 Click Close to exit the Dimension Style Manager.

Toolbar: Dimension or Styles
Command entry: DIMSTYLE

To add and format alternate units

1 Click Home tab ➤ Annotation panel ➤ Dimension Style.

2 In the Dimension Style Manager, select the style you want to change. Click Modify.

3 In the Modify Dimension Style dialog box, Alternate Units tab, select Display Alternate Units.

4 Under Alternate Units

- Select a unit format from the list.
- Select a precision value for the alternate units.
- Enter any prefix and suffix for the displayed dimension, including a space character if you want a gap between the dimension and the prefix or suffix.

5 Click OK.

6 Click Close to exit the Dimension Style Manager.

Toolbar: Dimension or Styles
Command entry: DIMSTYLE

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

System Variables

DIMALT

Controls the display of alternate units in dimensions.

DIMALTD

Controls the number of decimal places in alternate units.

DIMALTF

Controls the multiplier for alternate units.

DIMALTTD

Sets the number of decimal places for the tolerance values in the alternate units of a dimension.

DIMALTU

Sets the units format for alternate units of all dimension substyles except Angular.

DIMALTZ

Controls the suppression of zeros for alternate unit dimension values.

DIMAPOST

Specifies a text prefix or suffix (or both) to the alternate dimension measurement for all types of dimensions except angular.

DIMAUNIT

Sets the units format for angular dimensions.

DIMDEC

Sets the number of decimal places displayed for the primary units of a dimension.

DIMDSEP

Specifies a single-character decimal separator to use when creating dimensions whose unit format is decimal.

DIMLFAC

Sets a scale factor for linear dimension measurements.

DIMLUNIT

Sets units for all dimension types except Angular.

DIMPOST

Specifies a text prefix or suffix (or both) to the dimension measurement.

DIMTDEC

Sets the number of decimal places to display in tolerance values for the primary units in a dimension.

Utilities

No entries

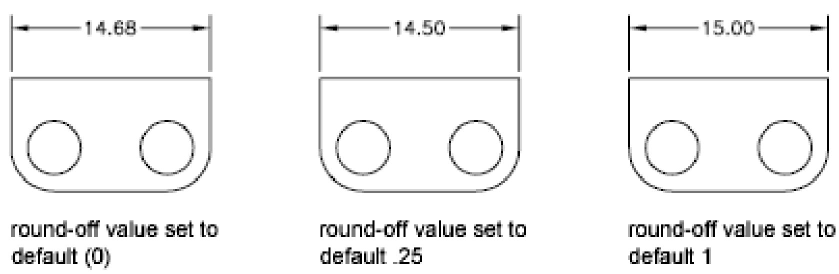
Command Modifiers

No entries

Round Off Dimension Values

You can round off the numeric values in dimensions and lateral tolerances.

You can round off all dimension values except those for angular dimensions. For example, if you specify a round-off value of 0.25, all distances are rounded to the nearest 0.25 unit. The number of digits displayed after the decimal point depends on the precision set for primary and alternate units and lateral tolerance values.



To round off dimension values

1 Click Home tab ➤ Annotation panel ➤ Dimension Style. 

2 In the Dimension Style Manager, select the style you want to change. Click Modify.

3 In Modify Dimension Style dialog box, Primary Units tab, under Linear Dimensions, enter the round-off value.

4 Click OK.

5 Click Close to exit the Dimension Style Manager.

Toolbar: Dimension or Styles
Command entry: DIMSTYLE

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

System Variables

DIMRND

Rounds all dimensioning distances to the specified value.

Utilities

No entries

Command Modifiers

No entries

Suppress Zeros in Dimensions

You can suppress leading and trailing zeros in the numeric portion of dimension text. You can also specify the sub unit for the dimension distance.

If you suppress leading zeros in decimal dimensions, 0.500 becomes .500. If you suppress trailing zeros, 0.500 becomes 0.5. You can suppress *both* leading and trailing zeros so that 0.5000 becomes .5 and 0.0000 becomes 0.

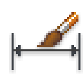
For dimension distances less than one unit, you can set the dimension distance to display in sub units. If the distance is shown in *m*, you can set to display distances less than one *m* in *cm* or *mm*.

The table shows the effect of selecting each option and provides examples of the architectural units style. If feet are included with a fractional inch, the number of inches is indicated as zero, no matter which option you select. Thus, the dimension 4'-3/4" becomes 4'-0 3/4".

Zero suppression for feet and inches

Option	Effect	Examples			
No options selected	Includes zero feet and zero inches	0'-0 1/2"	0'-6"	1'-0"	1'-0 3/4"
0 Inches selected	Suppresses zero inches (includes zero feet)	0'-0 1/2"	0'-6"	1'	1'-0 3/4"
0 Feet selected	Suppresses zero feet (includes zero inches)	1/2"	6"	1'-0"	1'-0 3/4"
0 Feet and 0 Inches selected	Suppresses zero feet and zero inches	1/2"	6"	1'	1'-0 3/4"

To suppress zeros in dimension values

1 Click Home tab ➤ Annotation panel ➤ Dimension Style. 

2 In the Dimension Style Manager, select the style you want to change. Click Modify.

3 In the Modify Dimension Style dialog box, Primary Units tab or Alternate Units tab, under Zero Suppression, select from the following:

- **Sub-units factor:** Suppresses leading zeros in decimal values.
- **Trailing.** Suppresses trailing zeros in decimal values.
- **0 Feet.** Suppresses display of 0 feet in feet and inches values.
- **0 Inches.** Suppresses display of 0 inches in feet and inches values.

4 Click OK.

5 Click Close to exit the Dimension Style Manager.

Toolbar: Dimension or Styles

Command entry: DIMSTYLE

To display dimension value in sub units

1. Click Home tab ➤ Annotation panel ➤ Dimension Style.
2. In the Dimension Style Manager, select the style you want to change. Click Modify.
3. In the Modify Dimension Style dialog box, Primary Units tab or Alternate Units tab, under Zero Suppression, select Leading.
   - **Sub-units factor:** Sets the number of sub units to a unit. It is used to display the dimension distance in a sub unit, for distances less than one unit. For example, enter **100** if the suffix is m and the sub-unit suffix is to display in cm.
   - **Sub-units suffix:** Includes a suffix to the dimension text sub unit. You can enter text or use control codes to display special symbols. For example, enter cm to for .96m to display as 96cm.
4. Click OK.
5. Click Close to exit the Dimension Style Manager.

Toolbar: Dimension or Styles

Command entry: DIMSTYLE

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

System Variables

DIMALTTZ

Controls suppression of zeros in tolerance values.

DIMALTZ

Controls the suppression of zeros for alternate unit dimension values.

DIMAZIN

Suppresses zeros for angular dimensions.

DIMTZIN

Controls the suppression of zeros in tolerance values.

DIMZIN

Controls the suppression of zeros in the primary unit value.

Utilities

No entries

Command Modifiers

No entries

Display Lateral Tolerances

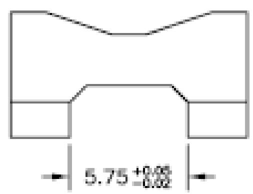
Lateral tolerances are values indicating the amount a measured distance can vary. You can control whether lateral tolerances are displayed and you can choose from several styles of lateral tolerances.

A lateral tolerance specifies the amount by which a dimension can vary. By specifying tolerances in manufacturing, you can control the degree of accuracy needed for a feature. A feature is some aspect of a part, such as a point, line, axis, or surface.

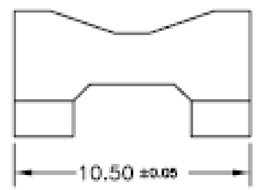
You can apply tolerances directly to a dimension by appending the tolerances to the dimension text. These dimension tolerances indicate the largest and smallest permissible size of the dimension. You can also apply geometric tolerances, which indicate deviations of form, profile, orientation, location, and runout.

Lateral tolerances can be specified from theoretically exact measurements. These are called basic dimensions and have a box drawn around them.

If the dimension value can vary in both directions, the plus and minus values you supply are appended to the dimension value as deviation tolerances. If the deviation tolerance values are equal, they are displayed with a   sign and they are known as symmetrical. Otherwise, the plus value goes above the minus value.

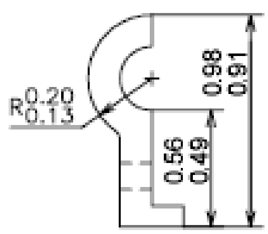


deviation tolerance



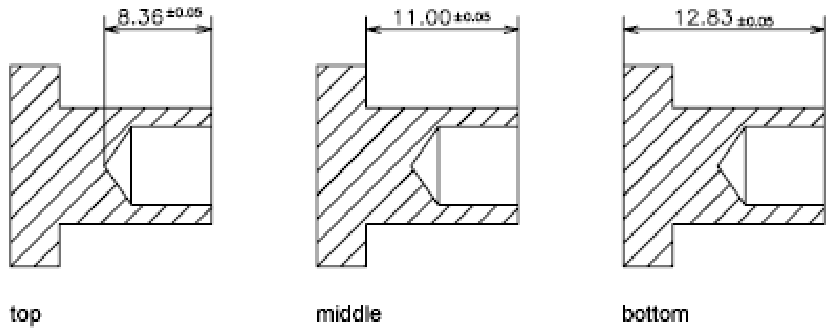
symmetrical deviation tolerance

If the tolerances are applied as limits, the program uses the plus and minus values you supply to calculate a maximum and minimum value. These values replace the dimension value. If you specify limits, the upper limit goes above the lower.

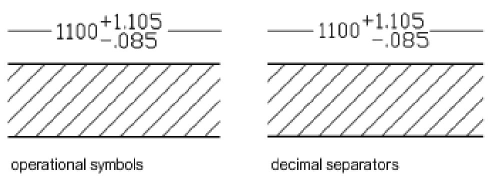


Format Lateral Tolerances

You can control the vertical placement of tolerance values relative to the main dimension text. Tolerances can align with the top, middle, or bottom of the dimension text.



Along with vertical placement of tolerance values, you can also control the horizontal alignment of the upper and lower tolerance values.The upper and lower tolerance values can be aligned using either the operational symbols or decimal separators.



You can also control zero suppression as you can with the primary and alternate units. Suppressing zeros in lateral tolerances has the same effect as suppressing them in the primary and alternate units. If you suppress leading zeros, 0.5 becomes .5, and if you suppress trailing zeros, 0.5000 becomes 0.5.

See also:

■ Add Geometric Tolerances on page 1736

To specify methods for lateral tolerances

1 Click Home tab ➤ Annotation panel ➤ Dimension Style. 

2 In the Dimension Style Manager, select the style you want to change. Click Modify.

3 In the Modify Dimension Style dialog box, Tolerances tab, under Tolerance Format, select a method from the Method list, and then do one of the following:
   - If you select Limits, enter upper and lower tolerance deviation in the Upper Value and Lower Value boxes.
   - If you select Symmetrical tolerances, Lower Value is not available, because you need only one tolerance value.
   - If you select Basic, enter a value in Offset from Dim Line (on the Text tab) to represent the gap between the text and its enclosing box.
4 Click OK.
5 Click Close to exit the Dimension Style Manager.

Toolbar: Dimension or Styles
Command entry: DIMSTYLE

To align and suppress zeros in tolerance values

1 Click Home tab ➤ Annotation panel ➤ Dimension Style.
2 In the Dimension Style Manager, select the style you want to change. Click Modify.
3 In the Modify Dimension Style dialog box, Tolerances tab, under Tolerance Format, select the alignment from the Vertical Position list.
4 To suppress zeros in primary or alternate units, under Zero Suppression, select Leading to suppress leading zeros. Select Trailing to suppress trailing zeros.
5 Click OK.
6 Click Close to exit the Dimension Style Manager.

Toolbar: Dimension or Styles
Command entry: DIMSTYLE

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

System Variables

DIMALTTD

Sets the number of decimal places for the tolerance values in the alternate units of a dimension.

DIMALTTZ

Controls suppression of zeros in tolerance values.

DIMGAP

Sets the distance around the dimension text when the dimension line breaks to accommodate dimension text.

DIMLIM

Generates dimension limits as the default text.

DIMTDEC

Sets the number of decimal places to display in tolerance values for the primary units in a dimension.

DIMTFAC

Specifies a scale factor for the text height of fractions and tolerance values relative to the dimension text height, as set by DIMTXT.

DIMTM

Sets the minimum (or lower) tolerance limit for dimension text when DIMTOL or DIMLIM is on.

DIMTOL

Appends tolerances to dimension text.

DIMTP

Sets the maximum (or upper) tolerance limit for dimension text when DIMTOL or DIMLIM is on.

DIMTZIN

Controls the suppression of zeros in tolerance values.

Utilities

No entries

Command Modifiers

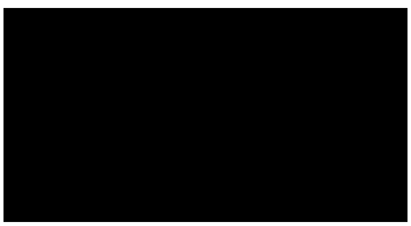
No entries

Control the Display of Fractions

You can control the format of the fraction displayed in dimensions.

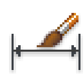
You can set the fraction format in dimensions using the DIMFRAC system variable when the DIMLUNIT system variable is set to 4(architecture) or 5 (fractional).

The following illustration shows the different fraction formats available.



These settings are available on the Primary Units tab of the Dimension Style Manager.

To specify the fraction format

1 Click Home tab ➤ Annotation panel ➤ Dimension Style. 

2 In the Dimension Style Manager, select the style you want to change. Click Modify.

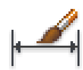
3 In the Modify Dimension Style dialog box, Primary Units tab, under Linear dimensions, select one of the following from Fraction format:

- Horizontal
- Diagonal

- Not Stacked

4 Click OK.

5 Click Close to exit the Dimension Style Manager.

Toolbar: Dimension or Styles 
Command entry: DIMSTYLE

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

System Variables

DIMFRAC

Sets the fraction format when DIMLUNIT is set to 4 (Architectural) or 5 (Fractional).

DIMLUNIT

Sets units for all dimension types except Angular.

Utilities

No entries

Command Modifiers

No entries

Set the Scale for Dimensions

You can specify the size of dimensions in your drawing. How you set dimension size depends on the method you use to lay out and plot drawings.

Dimension scale affects the size of the dimension geometry relative to the objects in the drawing. Dimension scale affects sizes, such as text height and arrowhead size, and offsets, such as the extension line origin offset. You should set these sizes and offsets to values that represent their actual plotted size. Dimension scale does *not* apply the overall scale factor to tolerances or measured lengths, coordinates, or angles.

NOTE You can use annotative scaling to control the overall scale of dimensions displayed in layout viewports. When you create annotative dimensions, they are scaled based on the current annotation scale setting and automatically displayed at the correct size.

Setting dimension scale depends on how you lay out your drawing. There are three methods used to create dimensions in a drawing layout:

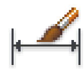
- **Dimension in model space for plotting in model space**. This is the traditional method used with single-view drawings. To create dimensions that are scaled correctly for plotting, set the DIMSCALE system variable to the inverse of the intended plot scale. For example, if the plot scale is 1/4, set DIMSCALE to 4.
- **Dimension in model space for plotting in paper space**. This was the preferred method for complex, multiple-view drawings prior to AutoCAD 2002. Use this method when the dimensions in a drawing need to be referenced by other drawings (xrefs) or when creating isometric dimensions in 3D isometric views. To prevent the dimensions in one layout viewport from being displayed in other layout viewports, create a dimensioning layer for each layout viewport that is frozen in all other layout viewports. To create dimensions that are scaled automatically for display in a paper space layout, set the DIMSCALE system variable to 0.
- **Dimension in layouts**. This is the simplest dimensioning method. Dimensions are created in paper space by selecting model space objects or by specifying object snap locations on model space objects. By default, associativity between paper space dimensions and model space objects is maintained. No additional scaling is required for dimensions created in a paper space layout: DIMLFAC and DIMSCALE do not need to be changed from their default value of 1.0000.

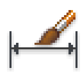
NOTE When you dimension model space objects in paper space using associative dimensions, dimension values for the display scale of each viewport are automatically adjusted. This adjustment is combined with the current setting for DIMLFAC and is reported by the LIST command as a dimension style override. For nonassociative dimensions, you must set DIMLFAC manually.

See also:

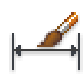
- Draw, Scale, and Annotate in Model Space on page 425
- Scale Views in Layout Viewports on page 456
- Scale Annotations on page 1393

To set the overall dimension scale

1. Click Home tab ➤ Annotation panel ➤ Dimension Style. 
2. In the Dimension Style Manager, select the style you want to change. Click Modify.
3. In Modify Dimension Style dialog box, Fit tab, under Scale for Dimension Features, enter a value for the overall scale.
4. Click OK.
5. Click Close to exit the Dimension Style Manager.

Toolbar: Dimension or Styles 
Command entry: DIMSTYLE

To set the dimension scale for model space dimensions in layouts

1. Click Home tab ➤ Annotation panel ➤ Dimension Style. 
2. In the Dimension Style Manager, select the style you want to change. Click Modify.
3. In Modify Dimension Style dialog box, Fit tab, under Scale for Dimension Features, select Scale Dimension to Layout (Paper space).
4. Click OK.
5. Click Close to exit the Dimension Style Manager.

Toolbar: Dimension or Styles 

Command entry: DIMSTYLE

To set dimension scale for creating dimensions in a layout

1 Click a layout tab to switch to paper space.

2 To create dimensions in paper space with the correct model space dimension values, use object snap modes to snap to points in model space from paper space or select the objects directly.

  The DIMLFAC system variable can be changed if you need to convert the linear dimension values between the imperial and metric measurement systems.

Quick Reference

Commands

DIMREGEN

Updates the locations of all associative dimensions.

DIMSTYLE

Creates and modifies dimension styles.

System Variables

DIMASSOC

Controls the associativity of dimension objects and whether dimensions are exploded.

DIMLFAC

Sets a scale factor for linear dimension measurements.

DIMSCALE

Sets the overall scale factor applied to dimensioning variables that specify sizes, distances, or offsets.

Utilities

No entries

Command Modifiers

No entries

Create Dimensions

You can create all of the standard types of dimensions.

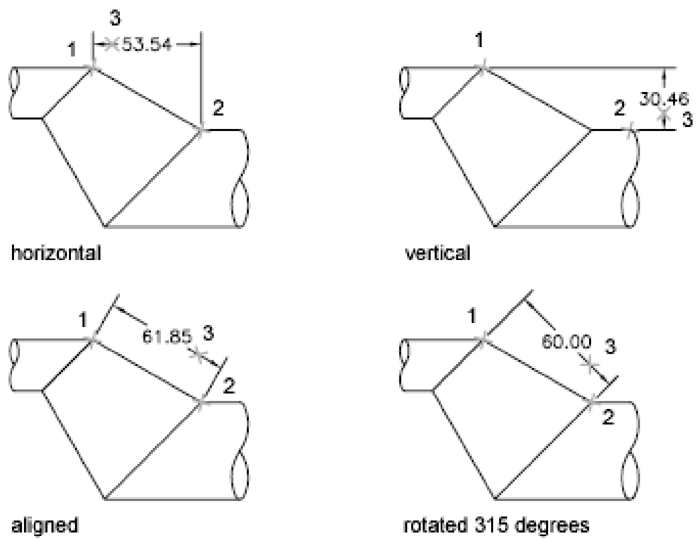
Create Linear Dimensions

You can create linear dimensions with horizontal, vertical, and aligned dimension lines. These linear dimensions can also be stacked, or they can be created end to end.

Overview of Creating Linear Dimensions

Linear dimensions can be horizontal, vertical, or aligned. With aligned dimensions, the dimension line is parallel to the line (imaginary or real) between the extension line origins. Baseline (or parallel) and continued (or chain) dimensions are series of consecutive dimensions that are based on a linear dimension.

In all four illustrations, the extension line origins are designated explicitly at 1 and 2, respectively. The dimension line location is specified at 3.



As you create linear dimensions, you can modify the content of the text, the angle of the text, or the angle of the dimension line.

Quick Reference

Commands

DIMALIGNED

Creates an aligned linear dimension.

DIMBASELINE

Creates a linear, angular, or ordinate dimension from the baseline of the previous or selected dimension.

DIMCONTINUE

Creates a dimension that starts from an extension line of a previously created dimension.

DIMEDIT

Edits dimension text and extension lines.

DIMLINEAR

Creates a linear dimension.

DIMSTYLE

Creates and modifies dimension styles.

QDIM

Quickly creates a series of dimensions from selected objects.

System Variables

DIMDLI

Controls the spacing of the dimension lines in baseline dimensions.

Utilities

No entries

Command Modifiers

No entries

Create Horizontal and Vertical Dimensions

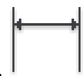
You can create dimensions using only the horizontal or vertical components of the locations or objects that you specify.

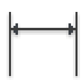
The program automatically applies a horizontal or vertical dimension according to the extension line origins that you specify or the location where you select an object; however, you can override this as you create the dimension by specifying that a dimension be horizontal or vertical. For example, in the following illustration, a horizontal dimension is drawn by default unless you specify a vertical one.

horizontal dimension created by default

vertical dimension specified

To create a horizontal or vertical dimension

1. Click Home tab ➤ Annotation panel ➤ Linear. 
2. Press ENTER to select the object to dimension, or specify the first and second extension line origins.
3. Before specifying the dimension line location, you can override the dimension direction and edit the text, the text angle, or the dimension line angle:
   - To rotate the extension lines, enter **r** (Rotated). Then enter the dimension line angle.
   - To edit the text, enter **m** (multiline text). In the In-place Text Editor, revise the text. Click OK.
     Editing within or overwriting the brackets (<>) changes or removes the dimension value calculated by the program. Adding text before or after the brackets appends text before or after the dimension value.
   - To rotate the text, enter **a** (Angle). Then enter the text angle.
4. Specify the dimension line location.

Toolbar: Dimension 
Command entry: DIMLINEAR

Quick Reference

Commands

DIMLINEAR

Creates a linear dimension.

System Variables

DIMEXO

Specifies how far extension lines are offset from origin points.

Utilities

No entries

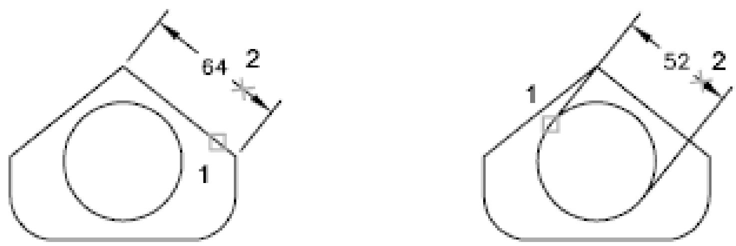
Command Modifiers

No entries

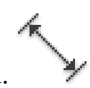
Create Aligned Dimensions

You can create dimensions that are parallel to the locations or objects that you specify.

In aligned dimensions, the dimension line is parallel to the extension line origins. The illustration shows two examples of aligned dimensioning. The object is selected (1), and the location of the aligned dimension is specified (2). The extension lines are drawn automatically.



To create an aligned dimension

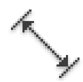
1 Click Home tab ➤ Annotation panel ➤ Aligned. 

2 Press ENTER to select the object to dimension, or specify the first and second extension line origins.

3 Before specifying the dimension line location, you can edit the text or change the text angle. Editing within or overwriting the brackets (<>) changes or removes the dimension value calculated by the program. Adding text before or after the brackets appends text before or after the dimension value.
   - To edit the text using multiline text, enter **m** (multiline text). In the In-place Text Editor, revise the text. Click OK.

- To edit the text using single-line text, enter **t** (Text). Revise the text at the command prompt and press ENTER.
- To rotate the text, enter **a** (Angle). Then enter the text angle.

4 Specify the dimension line location.

Toolbar: Dimension 
Command entry: DIMALIGNED

Quick Reference

Commands

DIMALIGNED

Creates an aligned linear dimension.

DIMSTYLE

Creates and modifies dimension styles.

System Variables

DIMEXO

Specifies how far extension lines are offset from origin points.

Utilities

No entries

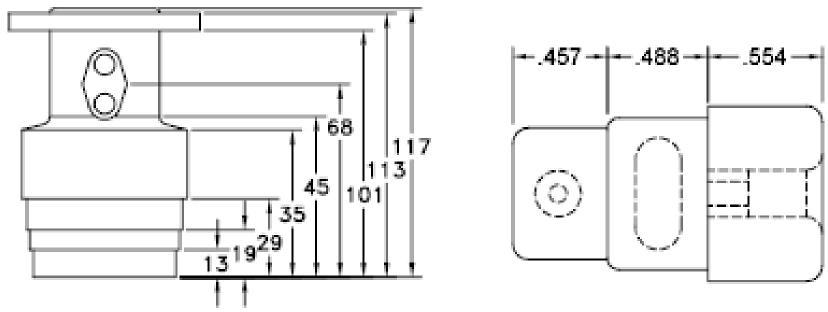
Command Modifiers

No entries

Create Baseline and Continued Dimensions

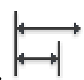
Baseline dimensions are multiple dimensions measured from the same baseline. Continued dimensions are multiple dimensions placed end to end.

You must create a linear, aligned, or angular dimension before you create baseline or continued dimensions. You create baseline dimensions incrementally from the most recently created dimension in the current session.



Both baseline and continued dimensions are measured from the previous extension line unless you specify another point as the point of origin.

To create a baseline linear dimension

1 Click Annotate tab ➤ Dimensions panel ➤ Baseline. 

   By default, the origin of the last linear dimension created is used as the first extension line for the new baseline dimension. You are prompted for the second dimension line.

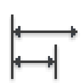
2 Use an object snap to select the second extension line origin, or press ENTER to select any dimension as the base dimension.

   The program automatically places the second dimension line at the distance specified by the Baseline Spacing option in the Dimension Style Manager, Lines tab.

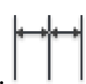
3 Use an object snap to specify the next extension line origin.

4 Continue to select extension line origins as required.

5 Press ENTER twice to end the command.

Toolbar: Dimension 
Command entry: DIMBASELINE

To create a continued linear dimension

1 Click Annotate tab ➤ Dimensions panel ➤ Continue. 

The program uses the origin of the second extension line of the existing dimension as the first extension line origin.

2 Use object snaps to specify additional extension line origins.

3 Press ENTER twice to end the command.

Toolbar: Dimension
Command entry: DIMCONTINUE

Quick Reference

Commands

DIMBASELINE

Creates a linear, angular, or ordinate dimension from the baseline of the previous or selected dimension.

DIMCONTINUE

Creates a dimension that starts from an extension line of a previously created dimension.

DIMSTYLE

Creates and modifies dimension styles.

System Variables

DIMDLI

Controls the spacing of the dimension lines in baseline dimensions.

Utilities

No entries

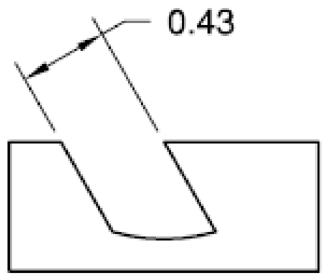
Command Modifiers

No entries

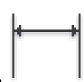
Create Rotated Dimensions

In rotated dimensions, the dimension line is placed at an angle to the extension line origins.

The illustration shows an example of a rotated dimension. In the example, the angle specified for dimension rotation is equal to the angle of the slot.



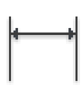
To create a rotated dimension

1 Click Home tab ➤ Annotation panel ➤ Linear. 

2 Press ENTER to select the object to dimension or specify the first and second extension line origins.

3 To rotate the dimension line, enter **r** (Rotated). Then enter the dimension line angle.

4 Specify the dimension line location.

Toolbar: Dimension 
Command entry: DIMLINEAR

Quick Reference

Commands

DIMALIGNED

Creates an aligned linear dimension.

DIMLINEAR

Creates a linear dimension.

System Variables

No entries

Utilities

No entries

Command Modifiers

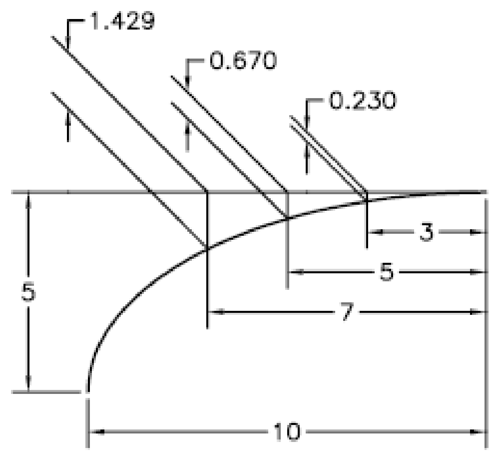
No entries

Create Dimensions with Oblique Extension Lines

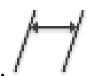
You can create dimensions with extension lines that are not perpendicular to their dimension lines.

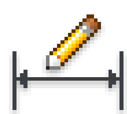
Extension lines are created perpendicular to the dimension line. However, if the extension lines conflict with other objects in a drawing, you can change their angle after the dimension has been drawn.

New dimensions are *not* affected when you make an existing dimension oblique.



To make extension lines oblique

1 Click Annotate tab ➤ Dimensions panel ➤ Oblique.. 
2 Select the dimension.
3 Enter a value for the angle of obliqueness, or specify two points.

Toolbar: Dimension 
Command entry: DIMEDIT

Quick Reference

Commands

DIMEDIT

Edits dimension text and extension lines.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

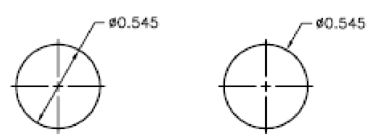
Create Radial Dimensions

Radial dimensions measure the radii and diameters of arcs and circles with optional centerlines or a center mark.

There are two types of radial dimensions:

- DIMRADIUS measures the radius of an arc or circle, and displays the dimension text with the letter *R* in front of it.

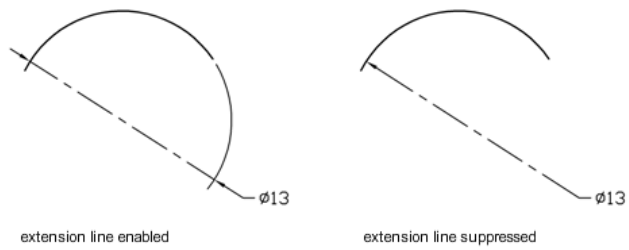
- DIMDIAMETER measures the diameter of an arc or circle, and displays the dimension text with the diameter symbol in front of it.



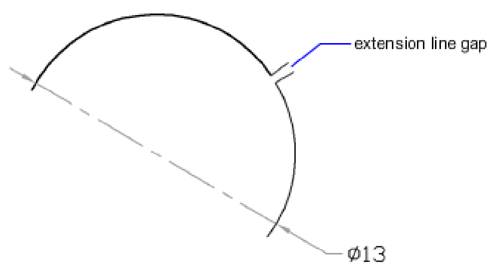
For horizontal dimension text, if the angle of the radial dimension line is greater than 15 degrees from horizontal, a hook line, also called a *dogleg* or *landing*, one arrowhead long, is created next to the dimension text.

Control Extension Lines

When an arc is dimensioned, the radial or diametric dimension does not have to be positioned along the arc directly. If a dimension is positioned past the end of an arc, either an extension line will be drawn that follows the path of the arc being dimensioned or no extension line will be drawn. When the extension line is suppressed (off), the dimension line of the radial or diametric dimension is drawn through the center point of the arc instead of to the extension line.



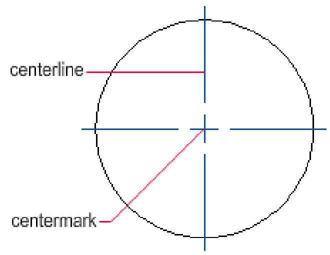
The DIMSE1 system variable controls whether or not a radial or diametric dimension will be drawn with an extension line when it is positioned off the end of an arc. When the display of the arc extension line is not suppressed, a gap between the arc and arc extension line is made. The size of the gap drawn is controlled with the DIMEXO system variable.



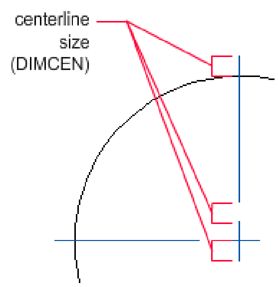
Control Centerlines and Center Marks

Depending on your dimension style settings, center marks and lines generate automatically for diameter and radius dimensions. They are created only if the dimension line is placed outside the circle or arc. You can create centerlines and center marks directly with the DIMCENTER command.

You can control the size and visibility of centerlines and center marks on the Modify Dimension Style dialog box, Symbols and Arrows tab, under Center Marks. You can also access this setting with the DIMCEN system variable.

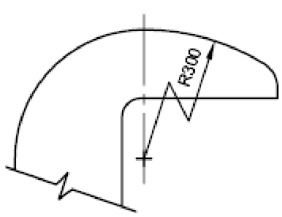


The size of the centerline is the length of the centerline segment that extends outside the circle or arc. It is also the size of the gap between the center mark and the start of the centerline. The size of the center mark is the distance from the center of the circle or arc to the end of the center mark.

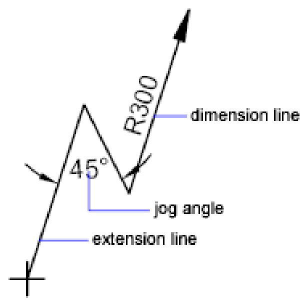


Create Jogged Radius Dimensions

With the DIMJOGGED command, you can create jogged radius dimensions, also called "foreshortened radius dimensions," when the center of an arc or circle is located off the layout and cannot be displayed in its true location. The origin point of the dimension can be specified at a more convenient location called the *center location override*.



You can control the default angle of the jog in the Modify Dimension Style dialog box, Symbols and Arrows tab, under Radius Dimension Jog.



Once a jogged radius dimension is created, you can modify the jog and the center location override by

- Using grips to move the features
- Changing the locations of the features with the Properties palette
- Using STRETCH

NOTE Jogged radius dimensions can be viewed but not edited in versions previous to AutoCAD 2006. Also, if you make dramatic changes to the associated geometry, you may get unpredictable results for the jogged radius dimension.

See also:

- Fit Dimension Text Within Extension Lines on page 1646

To create a diameter dimension

1. Click Home tab ➤ Annotation panel ➤ Diameter. 
2. Select the arc or circle to dimension.
3. Enter options as needed:
   - To edit the dimension text content, enter **t** (Text) or **m** (multiline text). Editing within or overwriting the brackets (<>) changes or removes the dimension value. Adding text before or after the brackets appends text before or after the dimension value.
   - To change the dimension text angle, enter **a** (Angle).
4. Specify the leader line location.

Toolbar: Dimension 
Command entry: DIMDIAMETER

To create a radius dimension

1 Click Home tab ➤ Annotation panel ➤ Radius. 

2 Select an arc, circle, or polyline arc segment.

3 Enter options as needed:
   - To edit the dimension text content, enter **t** (Text) or **m** (multiline text). Editing within or overwriting the brackets (<>) changes or removes the dimension value. Adding text before or after the brackets appends text before or after the dimension value.
   - To edit the dimension text angle, enter **a** (Angle).

4 Specify the leader line location.

Toolbar: Dimension 
Command entry: DIMRADIUS

To create a jogged radius dimension

1 Click Dimension menu ➤ Jogged. 

2 Select an arc, circle, or polyline arc segment.

3 Specify a point for the dimension origin (the center location override).

4 Specify a point for the dimension line angle and the dimension text location.

5 Specify another point for the location of the dimension jog.

Toolbar: Dimension 
Command entry: DIMJOGGED

To create centerlines automatically with radial dimensions

1 Click Home tab ➤ Annotation panel ➤ Dimension Style.

2 In the Dimension Style Manager, select the style you want to change. Click Modify.

3 In the Modify Dimension Style dialog box, Symbols and Arrows tab, under Center Marks, click Line.

4 In the Size box, enter the length of the centerline overshoot. Click OK.

5 Click Close to exit the Dimension Style Manager.

The example area in the dialog box displays the results of your changes.

Toolbar: Dimension or Styles
Command entry: DIMSTYLE

To create centerlines or center marks on an arc or circle

1 Click Home tab ➤ Annotation panel ➤ Dimension Style.

2 In the Dimension Style Manager, select the style you want to change. Click Modify.

3 In the Modify Dimension Style dialog box, Symbols and Arrows tab, under Center Marks, click Line.

4 In the Size box, enter the length of the centerline overshoot. Click OK.

5 Click Close to exit the Dimension Style Manager.

6 Click Annotate tab ➤ Dimensions panel ➤ Center Mark.

7 Select an arc or a circle.

Toolbar: Dimension
Command entry: DIMCENTER

To change the display arc extension line for radial or diametric dimensions

1 Select the radial or diametric dimension for which you want to suppress the arc extension line.
2 Right-click in the drawing. Click Properties.
3 In the Properties palette, Lines & Arrows category, click Ext Line.
4 Click the arrow next to Ext Line, and select On or Off from the list.
    - Select On to display the arc extension line.
    - Select Off to suppress the display of the arc extension line.
5 Press ESC to deselect the selected dimension.

Toolbar: Standard and Standard Annotation 
Command entry: PROPERTIES

Quick Reference

Commands

DIMCENTER

Creates the center mark or the centerlines of circles and arcs.

DIMDIAMETER

Creates a diameter dimension for a circle or an arc.

DIMJOGGED

Creates jogged dimensions for circles and arcs.

DIMRADIUS

Creates a radius dimension for a circle or an arc.

DIMSTYLE

Stores the name of the current dimension style.

QDIM

Quickly creates a series of dimensions from selected objects.

System Variables

DIMATFIT

Determines how dimension text and arrows are arranged when space is not sufficient to place both within the extension lines.

DIMCEN

Controls drawing of circle or arc center marks and centerlines by the DIMCENTER, DIMDIAMETER, and DIMRADIUS commands.

DIMEXO

Specifies how far extension lines are offset from origin points.

DIMJOGANG

Determines the angle of the transverse segment of the dimension line in a jogged radius dimension.

DIMJUST

Controls the horizontal positioning of dimension text.

DIMSE1

Suppresses display of the first extension line.

DIMTAD

Controls the vertical position of text in relation to the dimension line.

DIMTXTDIRECTION

Specifies the reading direction of the dimension text.

DIMTIH

Controls the position of dimension text inside the extension lines for all dimension types except Ordinate.

DIMTMOVE

Sets dimension text movement rules.

DIMTOFL

Controls whether a dimension line is drawn between the extension lines even when the text is placed outside.

DIMTOH

Controls the position of dimension text outside the extension lines.

DIMUPT

Controls options for user-positioned text.

Utilities

No entries

Command Modifiers

No entries

Create Angular Dimensions

Angular dimensions measure the angle between two lines or three points.

To measure the angle between two radii of a circle, you select the circle and specify the angle endpoints. With other objects, you select the objects and then specify the dimension location. You can also dimension an angle by specifying the angle vertex and endpoints. As you create the dimension, you can modify the text content and alignment before specifying the dimension line location.

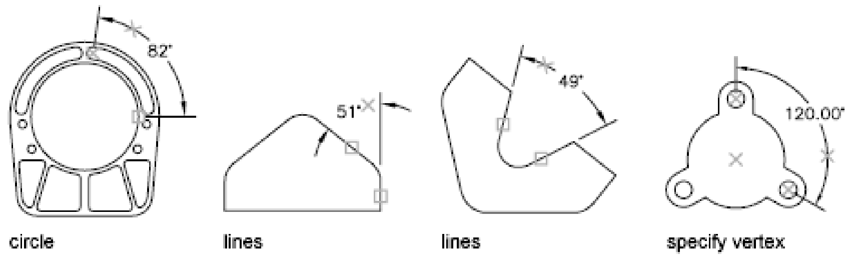
NOTE You can create baseline and continued angular dimensions relative to existing angular dimensions. Baseline and continued angular dimensions are limited to 180 degrees or less. To obtain baseline and continued angular dimensions larger than 180 degrees, use grip editing to stretch the location of the extension line of an existing baseline or continued dimension.

Dimension Lines

If you use two straight, nonparallel lines to specify an angle, the dimension line arc spans the angle between the two lines. If the dimension line arc does not meet one or both of the lines being dimensioned, The program draws one or two extension lines to intersect the dimension line arc. The arc is always less than 180 degrees.

Dimension Circles and Arcs

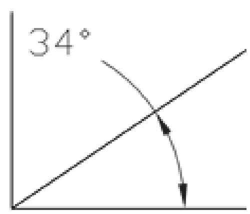
If you use an arc or a circle or three points to specify an angle, the program draws the dimension line arc between the extension lines. The extension lines are drawn from the angle endpoints to the intersection of the dimension line arc.



The location that you specify for the dimension line arc determines the quadrant of the dimensioned angle.

Dimension to a Quadrant

Angular dimensions can measure a specific quadrant that is formed when dimensioning the angle between of the endpoints of a line or arc, center point of a circle, or two vertices. As an angular dimension is being created, there are four possible angles that can be measured. By specifying a quadrant it allows you to ensure that the correct angle is dimensioned. When placing an angular dimension after a quadrant has been specified, you can place the dimension text outside of the extension lines of the dimension. The dimension line is automatically extended.

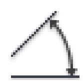


To create an angular dimension

1. Click Home tab ➤ Annotation panel ➤ Angular. 
2. Use one of the following methods:
   - To dimension a circle, select the circle at the first endpoint of the angle and then specify the second endpoint of the angle.
   - To dimension any other object, select the first line, and then select the second line.

3 Enter options as needed:
- To edit the dimension text content, enter **t** (Text) or **m** (multiline text). Editing within or overwriting the brackets (<>) changes or removes the calculated dimension value. Adding text before or after the brackets appends text before or after the dimension value.
- To edit the dimension text angle, enter **a** (Angle).
- To confine the dimension to a quadrant, enter **q** (Quadrant) and specify the quadrant to measure.

4 Specify the dimension line arc location.

Toolbar: Dimension 
Command entry: DIMANGULAR

Quick Reference

Commands

DIMANGULAR

Creates an angular dimension.

DIMBASELINE

Creates a linear, angular, or ordinate dimension from the baseline of the previous or selected dimension.

DIMCONTINUE

Creates a dimension that starts from an extension line of a previously created dimension.

System Variables

DIMADEC

Controls the number of precision places displayed in angular dimensions.

DIMAUNIT

Sets the units format for angular dimensions.

DIMDEC

Sets the number of decimal places displayed for the primary units of a dimension.

Utilities

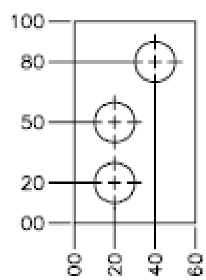
No entries

Command Modifiers

No entries

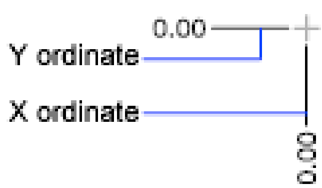
Create Ordinate Dimensions

Ordinate dimensions measure the perpendicular distance from an origin point called the *datum* to a feature, such as a hole in a part. These dimensions prevent escalating errors by maintaining accurate offsets of the features from the datum.



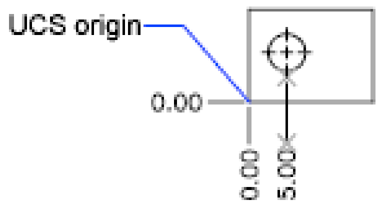
ordinate dimensions

Ordinate dimensions consist of an *X* or *Y* value with a leader line. *X*-datum ordinate dimensions measure the distance of a feature from the datum along the *X* axis. *Y*-datum ordinate dimensions measure the distance along the *Y* axis.



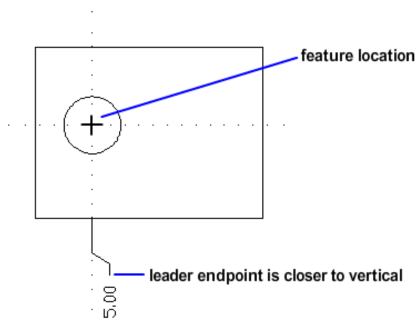
Locate the Datum

The location and orientation of the current UCS determines the ordinate values. Before creating ordinate dimensions, you typically set the UCS origin to coincide with the datum.



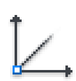
Locate the Leader

After you specify the feature location, you are prompted for the leader endpoint. By default, the leader endpoint that you specify automatically determines whether an *X*- or a *Y*-datum ordinate dimension is created. For example, you can create an *X*-datum ordinate dimension by specifying a location for the leader endpoint that is closer to vertical than horizontal.



After creating an ordinate dimension, you can easily relocate the dimension leader and text using grip editing. The dimension text is always aligned with the ordinate leader line.

To create ordinate dimensions

1 Click View tab ➤ Coordinates panel ➤ Origin. 

2 At the Specify New Origin Point prompt, specify an origin point.

   The origin point specified is used to define the value assigned to the ordinate dimension. Typically, the origin point is defined on the model.

3 Click Home tab ➤ Annotation panel ➤ Ordinate. 

4 If straight ordinate leaders are required, turn Ortho mode on.

5 At the Select Feature Location prompt, specify a point location.

6 Enter **x** (X Datum) or **y** (Y Datum).

   You can skip this step by making sure that the ordinate leader endpoint is close to vertical for an *X* datum or close to horizontal for a *Y* datum.

7 Specify the ordinate leader endpoint.

Toolbar: Dimension 
Command entry: DIMORDINATE

Quick Reference

Commands

DIMORDINATE

Creates ordinate dimensions.

QDIM

Quickly creates a series of dimensions from selected objects.

UCS

Manages user coordinate systems.

System Variables

No entries

Utilities

No entries

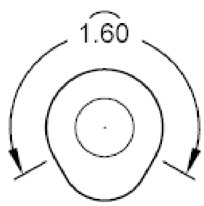
Command Modifiers

No entries

Create Arc Length Dimensions

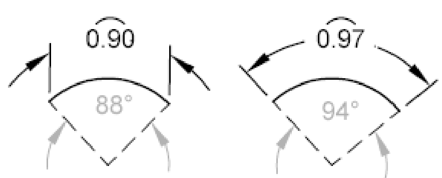
Arc length dimensions measure the distance along an arc or polyline arc segment.

Typical uses of arc length dimensions include measuring the travel distance around a cam or indicating the length of a cable. To differentiate them from linear or angular dimensions, arc length dimensions display an arc symbol by default.



The arc symbol, also called a hat or cap, is displayed either above the dimension text or preceding the dimension text. You can specify the placement style using the Dimension Style Manager. The placement style can be changed on the Symbols and Arrows tab of either the New Dimension Style dialog box or the Modify Dimension Style dialog box.

The extension lines of an arc length dimension can be orthogonal or radial.



NOTE Orthogonal extension lines are displayed only when the included angle of the arc is less than 90 degrees.

To create an arc length dimension

1. Click Home tab ➤ Annotation panel ➤ Arc Length.
2. Select an arc or polyline arc segment.
3. Specify the dimension line location.

Toolbar: Dimension
Command entry: DIMARC

Quick Reference

Commands

DIMARC

Creates an arc length dimension.

DIMSTYLE

Creates and modifies dimension styles.

PROPERTIES

Controls properties of existing objects.

System Variables

DIMARCSYM

Controls display of the arc symbol in an arc length dimension.

Utilities

No entries

Command Modifiers

No entries

Modify Existing Dimensions

You can modify all components of the existing dimension objects in a drawing either individually or by using dimension styles.

Apply a New Dimension Style to Existing Dimensions

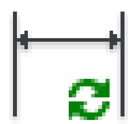
You can modify existing dimensions by applying a different dimension style. If you make changes to a dimension style, you can choose whether to update the dimensions associated with that dimension style.

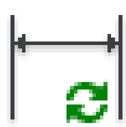
When you create a dimension, the current dimension style is associated with that dimension. The dimension retains this dimension style unless you apply a new dimension style to it or set up dimension style overrides.

You can modify existing dimensions by applying a different dimension style. If you make changes to a dimension style, you can choose whether to update the dimensions associated with that dimension style.

You can restore an existing dimension style or apply the current dimension style, including any dimension style overrides, to selected dimensions.

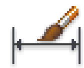
To apply the current dimension style to existing dimensions

1. Click Annotate tab ➤ Dimensions panel ➤ Update. 
2. Select the dimensions to update to the current dimension style.
3. Press ENTER.

Toolbar: Dimension, Update 

Command entry: DIMSTYLE

To restore a dimension style

1 Click Home tab ➤ Annotation panel ➤ Dimension Style. 

2 In the Dimension Style Manager, select the dimension style to restore. Click Set Current.

3 Click Close.

Toolbar: Dimension 
Command entry: DIMSTYLE

Alternate

- On the Styles toolbar, click the arrow in the Dimension Styles control and select a dimension style.

Quick Reference

Commands

DIMOVERRIDE

Controls overrides of system variables used in selected dimensions.

DIMSTYLE

Creates and modifies dimension styles.

PROPERTIES

Controls properties of existing objects.

System Variables

DIMCLRD

Assigns colors to dimension lines, arrowheads, and dimension leader lines.

Utilities

No entries

Command Modifiers

No entries

Override a Dimension Style

With dimension style overrides, you can temporarily change a dimensioning system variable without changing the current dimension style.

A dimension style override is a change made to specific settings in the current dimension style. It is equivalent to changing a dimensioning system variable without changing the current dimension style.

You can define dimension style overrides for individual dimensions, or for the current dimension style.

- For individual dimensions, you may want to create overrides to suppress a dimension's extension lines or modify text and arrowhead placement so that they do not overlap drawing geometry without creating a different dimension style.
- You can also set up overrides to the current dimension style. All dimensions you create in the style include the overrides until you delete the overrides, save the overrides to a new style, or set another style current. For example, if you choose Override in the Dimension Style Manager, and change the color of extension lines on the Lines tab, the current dimension style remains unchanged. However, the new value for color is stored in the DIMCLRE system variable. The next dimension you create will have extension lines in the new color. You can save the dimension style overrides as a new dimension style.

Some dimension characteristics are common to a drawing or to a style of dimensioning and are therefore suited to be permanent dimension style settings. Others generally apply on an individual basis and can be applied more effectively as overrides. For example, a drawing usually uses a single type of arrowhead, so it makes sense to define the arrowhead type as part of the dimension style. Suppression of extension lines, however, usually applies in individual cases only and is more suited to a dimension style override.

There are several ways to set up dimension style overrides. You can change options in the dialog boxes or change system variable settings at the command prompt. You reverse the override by returning the changed settings to their original values. The overrides apply to the dimension you are creating and all subsequent dimensions created with that dimension style until you reverse the override or make another dimension style current.

Example: Change a Dimension Style Override at the Command Prompt

You can override the current dimension style while creating a dimension by entering the name of any dimensioning system variable at any prompt. In this example, the dimension line color is changed. The change affects subsequent dimensions you create until you reverse the override or make another dimension style current.

Command: **dimoverride**
Enter dimension variable name to override or [Clear overrides]: **dimclrd**
Enter new value for dimension variable <BYBLOCK>: **5**
Enter dimension variable name to override: *Enter another dimension variable name or press* ENTER
Select objects: *Use an object selection method and press* ENTER *when you finish*

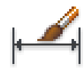
To set up dimension style overrides

1. Click Home tab ➤ Annotation panel ➤ Dimension Style.
2. In the Dimension Style Manager, under Styles, select the dimension style for which you want to create an override. Click Override.
3. In the Override Current Style dialog box, make changes to the dimension style by clicking the appropriate tab.
4. Click OK to return to the Dimension Style Manager.

   The dimension style overrides are listed below the style they modify in the dimension style name list.
5. Click Close.

Toolbar: Dimension or Styles
Command entry: DIMSTYLE

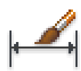
To apply dimension style overrides

1 Click Home tab ➤ Annotation panel ➤ Dimension Style. 

2 In the Dimension Style Manager, click Override.

3 In the Override Current Style dialog box, enter the style overrides. Click OK.

The program displays <style overrides> below the dimension style name in the Dimension Style Manager dialog box. After you create dimension style overrides, you can continue to modify dimension styles, compare them with other dimension styles, or delete or rename the overrides.

Toolbar: Dimension or Styles 
Command entry: DIMSTYLE

Quick Reference

Commands

DIMOVERRIDE

Controls overrides of system variables used in selected dimensions.

DIMSTYLE

Creates and modifies dimension styles.

PROPERTIES

Controls properties of existing objects.

System Variables

DIMCLRD

Assigns colors to dimension lines, arrowheads, and dimension leader lines.

Utilities

No entries

Command Modifiers

No entries

Modify A Dimension

Dimensions can be modified to include more information than just the values of the dimension. Dimensions can also be modified visually by using breaks and by adjusting the spacing between them.

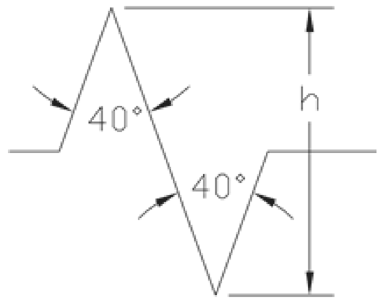
After you place a dimension, there are times when you need to modify the information that the dimension represents. You can add a jog line to a linear dimension to indicate that the dimension value does not represent the actual dimensioned value or add an inspection dimension to represent how often a dimension value of a manufactured part should be checked.

At times you might want to modify a dimension to simply improve readability. You can make sure that the extension or dimension lines do not obscure any objects; you can adjust the placement of linear dimensions so they are evenly spaced.

Dimension Jog

Jog lines can be added to linear dimensions. Jog lines are used to represent a dimension value that does not display the actual measurement. Typically, the actual measurement value of the dimension is smaller than the displayed value.

The jog is made up of two parallel lines and a cross line that forms two 40-degree angles. The height of the jog is determined by the linear jog size value of the dimension style.



Once you add a jog to a linear dimension, you can position it by using grips. To reposition the jog, select the dimension and then select the grip. Move the grip to another point along the dimension line. You can also adjust the height of the jog symbol on a linear dimension on the Properties palette under Lines & Arrows.

To add a jog to a linear dimension

1 Click Home tab ➤ Annotation panel ➤ Jog Line. 

2 Select a linear dimension.

3 Specify a point on the dimension line to place the jog.

Toolbar: Dimension 
Command entry: DIMJOGLINE

To add a jog to a linear dimension based on the midpoint of the selected dimension line

1 Click Home tab ➤ Annotation panel ➤ Jog Line. 

2 Select a linear dimension.

3 Press ENTER to position the jog at the midpoint of the selected dimension line.

Toolbar: Dimension 

Command entry: DIMJOGLINE

To reposition a jog using grips

1 With no command active, select the linear dimension that has the jog you want to reposition.

2 Select the grip in the middle of the jog.

The selected grip is highlighted, and the default grip mode, Stretch, is active.

3 Drag the crosshairs along the dimension line and click to reposition the jog.

TIP If you want to place the jog along the dimension line without changing the position of the dimension line, turn Ortho mode on.

Toolbar: Dimension 
Command entry: DIMJOGLINE

To remove a jog

1 Click Home tab ➤ Annotation panel ➤ Jog Line. 

2 Enter **r** (Remove) and press ENTER.

3 Select the linear dimension to remove the jog from.

Toolbar: Dimension 
Command entry: DIMJOGLINE

To modify the height of a jog using the Properties palette

1 With no command active, select the linear dimension with the jog whose height you want to change.

2 Right-click over the drawing window. Click Properties.

3 On the Properties palette, expand Lines & Arrows.

4 Select Jog Height Factor, and enter a new height for the jog.

5 Click outside the Properties palette. Press ESC.

Toolbar: Dimension
Command entry: DIMJOGLINE

Quick Reference

Commands

DIMALIGNED

Creates an aligned linear dimension.

DIMBASELINE

Creates a linear, angular, or ordinate dimension from the baseline of the previous or selected dimension.

DIMCONTINUE

Creates a dimension that starts from an extension line of a previously created dimension.

DIMJOGLINE

Adds or removes a jog line on a linear or aligned dimension.

DIMLINEAR

Creates a linear dimension.

DIMSTYLE

Creates and modifies dimension styles.

QDIM

Quickly creates a series of dimensions from selected objects.

System Variables

No entries

Utilities

No entries

Command Modifiers

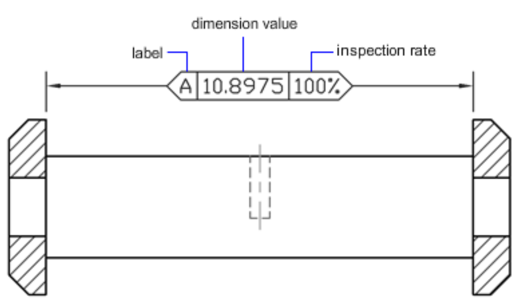
No entries

Inspection 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.

When working with parts that need to met a specific tolerance or dimension value before installing them into the final assembled product, you can use an inspection dimension to specify how often the part should be tested.

You can add an inspection dimension to any type of dimension object; it is composed of a frame and text values. The frame for an inspection dimension is made up of two parallel lines and the end is round or square. The text values are separated by vertical lines. An inspection dimension can contain up to three different fields of information: inspection label, dimension value, and inspection rate.



Inspection Dimension Fields

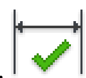
Inspection Label Text used to identify individual inspection dimensions. The label is located in the leftmost section of the inspection dimension.

Dimension Value Dimension value that is displayed is the same value before the inspection dimension is added. The dimension value can contain tolerances, text (both prefix and suffix), and the measured value. The dimension value is located in the center section of the inspection dimension.

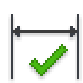
Inspection Rate Text used to communicate the frequency that the dimension value should be inspected, expressed as a percentage. The rate is located in the rightmost section of the inspection dimension.

You can add inspection dimensions to any type of dimension. The current values of an inspection dimension are displayed on the Properties palette, under Misc. The values include the properties that are used to control the look of the frame, and the text for both the label and rate values.

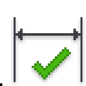
To create an inspection dimension

1. Click Annotate tab ➤ Dimensions panel ➤ Inspect. 
2. In the Inspection Dimension dialog box, click Select Dimensions.

   The Inspection Dimension dialog box closes. You are prompted to select dimensions.
3. Select the dimension you want to make an inspection dimension. Press ENTER to return to the dialog box.
4. Under the Shape section, specify the frame type.
5. Under the Label/Inspection rate section, specify the desired options.
   - Select the Label check box, and enter the desired label in the text box.
   - Select the Inspection Rate check box, and enter the desired rate in the text box.
6. Click OK.

Toolbar: Dimension 
Command entry: DIMINSPECT

To modify an inspection dimension from the Inspection dialog box

1. Click Annotate tab ➤ Dimensions panel ➤ Inspect. 
2. In the Inspection Dimension dialog box, click Select Dimensions.

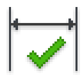
The Inspection Dimension dialog box closes. You are prompted to select dimensions.

3 Select the inspection dimension you want to modify. Press ENTER to return to the dialog box.

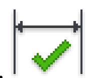
4 Under the Shape section, make the desired changes to the frame type.

5 Under the Label/Inspection rate section, make the desired changes to the label and inspection rate.

6 Click OK.

Toolbar: Dimension 
Command entry: DIMINSPECT

To remove an inspection dimension

1 Click Annotate tab ➤ Dimensions panel ➤ Inspect. 

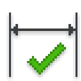
2 In the Inspection Dimension dialog box, click Select Dimensions.

The Inspection Dimension dialog box closes. You are prompted to select dimensions.

3 Select the dimension you want to remove the inspection dimension from. Press ENTER to return to the dialog box.

4 Click Remove Inspection.

5 Click OK.

Toolbar: Dimension 
Command entry: DIMINSPECT

To modify an inspection dimension using the Properties palette

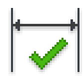
1 With no command active, select the inspection dimension you want to modify.

2 Right-click over the drawing window. Click Properties.

3 On the Properties palette, double-click the Misc caption of the pane to expand it.

4 Specify the new values for the Inspection shape, label, and rate.

5 Click outside the Properties palette. Press ESC.

Toolbar: Dimension 
Command entry: DIMINSPECT

Quick Reference

Commands

DIMINSPECT

Adds or removes inspection information for a selected dimension.

System Variables

No entries

Utilities

No entries

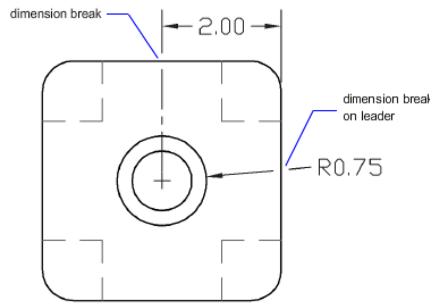
Command Modifiers

No entries

Dimension Breaks

With dimension breaks, you can keep the dimension, extension, or leader lines from appearing as if they are a part of the design.

Dimension breaks can be added to a dimension or a multileader automatically or manually. The method that you choose to place dimension breaks depends on the number of objects that intersect a dimension or multileader.



You can add dimension breaks to the following dimension and leader objects:

- Linear dimensions (aligned and rotated)
- Angular dimensions (2- and 3-point)
- Radial dimensions (radius, diameter, and jogged)
- Arc length dimensions
- Ordinate dimensions
- Multileaders (straight only)

The following dimension and leader objects do not support dimension breaks:

- Multileaders (spline only)
- “Legacy” leaders (straight or spline)

The following table explains the conditions where dimension breaks do not work or are not supported.

Dimension Break Exceptions

Condition	Description
No break in xrefs or blocks	Dimension breaks on dimensions or multileaders in xrefs and blocks are not supported. However, the objects in an xref or block can be used as the cutting edges for dimension breaks on dimensions or multileaders that are not in an xref or block.
No break on arrow-head and dimension text	Dimension breaks cannot be placed on an arrowhead or the dimension text. If you want a break to appear at the dimension text, it is recommended to use the background

Dimension Break Exceptions	
Condition	**Description**
	mask option. If the intersecting point of an object and the dimension are at the arrowhead or dimension text, the break will not be displayed until the intersecting object, or dimension or multileader are moved.
No break on trans-spatial dimensions	Automatic breaks are not supported for objects and dimensions or multileaders that are in different spaces. In order to break a dimension or multileader that is in a different space, you need to use the Manual option of the DIMBREAK command.

You can move dimension breaks from dimensions or multileaders. When removing dimension breaks from a dimension or multileader, all dimension breaks are removed. If there are some dimension breaks that you don't want to remove, you need to add them again.

The following objects can be used as cutting edges when adding a dimension break:

- Dimension
- Leader
- Line
- Circle
- Arc
- Spline
- Ellipse
- Polyline
- Text
- Multiline text
- Blocks but limited to the previously mentioned objects in this list
- Xrefs but limited to the previously mentioned objects in this list

Automatic Dimension Breaks

To create an automatically placed dimension break, you select a dimension or multileader, and then use the Auto option of the DIMBREAK command. Automatic dimension breaks are updated any time the dimension or multileader, or intersecting objects are modified.

You control the size of automatically placed dimension breaks on the Symbols and Arrows tab of the Dimension Style dialog box. The specified size is affected by the dimension break size, dimension scale, and current annotation scale for the current viewport. For more information about annotation scaling, see Scale Annotations on page 1393.

Dimension Break Created by Selecting an Object

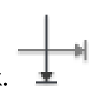
Instead of placing a dimension break for each object that intersects a dimension or multileader, you can specify which of the intersecting objects to use. Dimension breaks that are added by selecting individual intersecting objects are updated any time the dimension or multileader, or intersecting objects are modified.

Dimension Break Created by Picking Two Points

You can place a dimension break by picking two points on the dimension, extension, or leader line to determine the size and placement of the break. Dimension breaks that are added manually by picking two points are not automatically updated if the dimension or multileader, or intersecting object is modified.

So if a dimension or multileader with a manually added dimension break is moved or the intersecting object is modified, you might have to restore the dimension or multileader, and then add the dimension break again. The size of a dimension break that is created by picking two points is not affected by the current dimension scale or annotation scale value for the current viewport.

To automatically create dimension breaks for each intersecting object

1. Click Annotate tab ➤ Dimensions panel ➤ Break. 
2. Select a dimension or multileader.
3. Enter **a** (Auto) and press ENTER.

Toolbar: Dimension
Command entry: DIMBREAK

To create a single dimension break based on an intersecting object

1 Click Annotate tab ➤ Dimensions panel ➤ Break.

2 Select a dimension or multileader.

3 Select an object that intersects the dimension or multileader. Press ENTER.

Toolbar: Dimension
Command entry: DIMBREAK

To create a manual dimension break

1 Click Annotate tab ➤ Dimensions panel ➤ Break.

2 Select a dimension or multileader.

3 Enter **m** (Manual) and press ENTER.

4 Specify the first point on the dimension, extension, or leader line for the dimension break.

5 Specify the second point along the dimension, extension, or leader line for the dimension break.

Toolbar: Dimension
Command entry: DIMBREAK

To create dimension breaks for multiple dimensions or multileaders at one time

1 Click Annotate tab ➤ Dimensions panel ➤ Break.

2 Enter **m** (Multiple) and press ENTER.

3 Select the dimensions or multileaders to which to add the dimension breaks.

4 Enter **a** (Auto) and press ENTER.

Toolbar: Dimension
Command entry: DIMBREAK

To remove all dimension breaks from a dimension or multileader

1 Click Annotate tab ➤ Dimensions panel ➤ Break.

2 Select a dimension or multileader.

3 Enter **r** (Remove) and press ENTER.

Toolbar: Dimension
Command entry: DIMBREAK

To remove all dimension breaks from multiple dimensions or multileaders

1 Click Annotate tab ➤ Dimensions panel ➤ Break.

2 Enter **m** (Multiple), and press ENTER.

3 Select the dimensions or multileaders from which to remove the dimension breaks, and press ENTER.

4 Enter **r** (Remove), and press ENTER.

Toolbar: Dimension
Command entry: DIMBREAK

Quick Reference

Commands

DIMBREAK

Breaks or restores dimension and extension lines where they cross other objects.

DIMSTYLE

Creates and modifies dimension styles.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Adjust Dimension Spacing

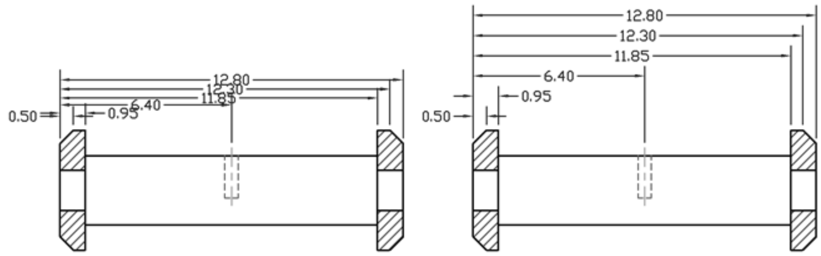
You can automatically adjust existing parallel linear and angular dimensions in a drawing so they are equally spaced or aligned at the dimension line with each other.

Parallel linear and angular dimensions can be created in a number of different ways in a drawing. With the DIMLINEAR and DIMANGULAR commands you can place one dimension at a time; you can use the DIMBASELINE and DIMCONTINUE commands to help place additional linear dimensions based on the previous linear dimension placed.

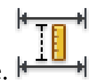
The DIMBASELINE command uses the DIMDLI system variable to create equally spaced dimensions, but once the dimensions are placed, changing the value of the system variable has no affect on the spacing of dimensions. If you change the text size or adjust the scale for the dimensions, they remain in the original position which can cause problems with overlapping dimension lines and text.

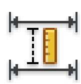
You can space linear and angular dimensions that overlap or are not equally spaced with the DIMSPACE command. The dimensions that are selected must be linear or angular, of the same type (rotated or aligned), parallel or concentric to one another, and on the extension lines of each other. You can also align linear and angular dimensions by using a spacing value of 0.

The following illustration shows parallel linear dimensions that are not equally spaced and then those that are equally spaced after using the DIMSPACE command.

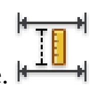


To equally space parallel linear and angular dimensions automatically

1 Click Annotate tab ➤ Dimensions panel ➤ Adjust Space. 
2 Select the dimension that you want to use as the base dimension when equally spacing dimensions.
3 Select the next dimension to equally space.
4 Continue to select dimensions and then press ENTER.
5 Enter **a** (Auto) and press ENTER.

Toolbar: Dimension 
Command entry: DIMSPACE

To equally space parallel linear and angular dimensions based on a distance

1 Click Annotate tab ➤ Dimensions panel ➤ Adjust Space. 
2 Select the dimension that you want to use as the base dimension when equally spacing dimensions.
3 Select the next dimension to equally space.
4 Continue to select dimensions and then press ENTER.

5 Enter a spacing value and press ENTER.

Toolbar: Dimension
Command entry: DIMSPACE

To align parallel linear and angular dimensions

1 Click Annotate tab ➤ Dimensions panel ➤ Adjust Space.

2 Select the dimension that you want to use as the base dimension when equally spacing dimensions.

3 Select the next dimension to align.

4 Continue to select dimensions and then press ENTER.

5 Enter **0** and press ENTER.

Toolbar: Dimension
Command entry: DIMSPACE

Quick Reference

Commands

DIMALIGNED

Creates an aligned linear dimension.

DIMANGULAR

Creates an angular dimension.

DIMBASELINE

Creates a linear, angular, or ordinate dimension from the baseline of the previous or selected dimension.

DIMCONTINUE

Creates a dimension that starts from an extension line of a previously created dimension.

DIMLINEAR

Creates a linear dimension.

DIMSPACE

Adjusts the spacing between linear dimensions or angular dimensions.

DIMSTYLE

Creates and modifies dimension styles.

QDIM

Quickly creates a series of dimensions from selected objects.

System Variables

DIMDLI

Controls the spacing of the dimension lines in baseline dimensions.

Utilities

No entries

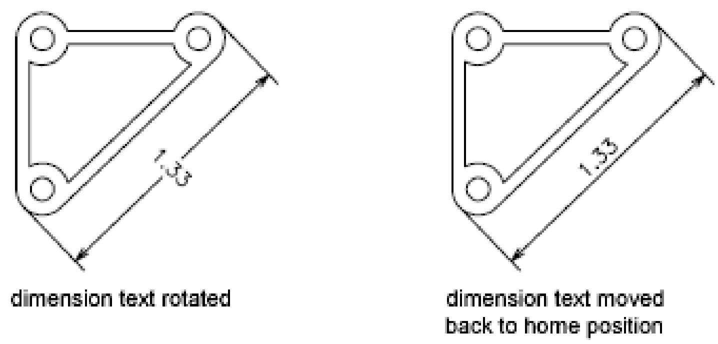
Command Modifiers

No entries

Modify Dimension Text

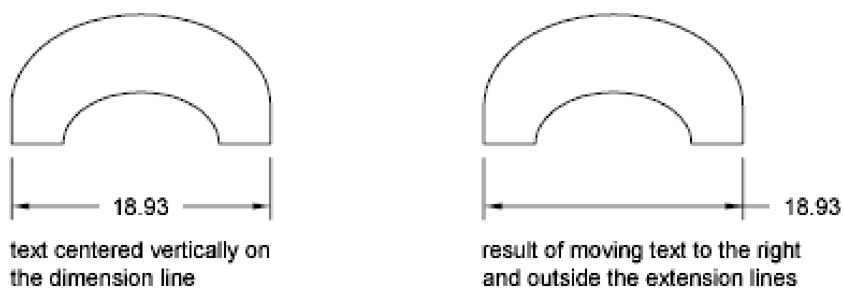
Once you've created a dimension, you can change the location and orientation of the existing dimension text or replace it with new text.

Once you've created a dimension, you can rotate the existing text or replace it with new text. You can move the text to a new location or back to its home position, which is the position defined by the current dimension style. In the following illustration, the home position is above and centered on the dimension line.



When you rotate or replace dimension text, you specify the change first, for example, rotating the text to be at an angle. When you move dimension text, you select a single dimension to move.

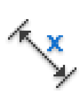
You can move dimension text to the left, right, or center along the dimension line or to any position inside or outside the extension lines. A quick and simple way to do this is by using grips. If you move text up or down, the current vertical alignment of the text relative to the dimension line is not changed, so the dimension and extension lines are modified accordingly. The following illustration shows the result of moving text down and to the right. The text remains centered vertically in relation to the dimension line.



See also:

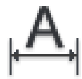
■ Control Dimension Text on page 1646

To rotate dimension text

1 tab ➤ panel ➤ . 

2 Select the dimension to edit.

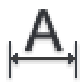
3 Enter the new angle for the text.

Toolbar: Dimension 
Command entry: DIMTEDIT

To return dimension text to its home position

1 Click Dimension menu ➤ Align Text ➤ Home.

2 Select the dimension text you want to return to its home position.

Toolbar: Dimension 
Command entry: DIMTEDIT

To replace existing dimension text with new text

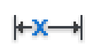
1 Click Modify menu ➤ Object ➤ Text ➤ Edit.

2 Select the dimension text you want to edit.

3 In the In-Place Text Editor, enter the new dimension text. Click OK.

Toolbar: Text 
Command entry: DDEDIT

To move text to the left side of the dimension line

1 Click Annotate tab ➤ Dimensions panel ➤ Left Justify. 

2 Select the dimension.

The dimension text is left-justified along the dimension line inside the extension lines. You can choose the Center or Right options to move the text to the center or right of the dimension line.

Toolbar: Dimension
Command entry: DIMTEDIT

To set dimension line spacing for baseline and continued dimensions

1 Click Home tab ➤ Annotation panel ➤ Dimension Style.

2 In the Dimension Style Manager, select the style you want to change. Click Modify.

3 In the Modify Dimension Style dialog box, Lines tab, under Dimension Lines, enter in the Baseline Spacing box the offset distance between dimension lines for baseline and continued dimensions.

4 Click OK.

5 Click Close to exit the Dimension Style Manager.

Toolbar: Dimension or Styles
Command entry: DIMSTYLE

To change the extension origin offset

1 Click Home tab ➤ Annotation panel ➤ Dimension Style.

2 In the Dimension Style Manager, select the style you want to change. Click Modify.

3 In the Modify Dimension Style dialog box, Lines tab, under Extension Lines, enter the new value for Offset from Origin.

4 Click OK.

5 Click Close to exit the Dimension Style Manager.

Toolbar: Dimension or Styles
Command entry: DIMSTYLE

Quick Reference

Commands

DDEDIT

Edits single-line text, dimension text, attribute definitions, and feature control frames.

DIMEDIT

Edits dimension text and extension lines.

DIMTEDIT

Moves and rotates dimension text and relocates the dimension line.

PROPERTIES

Controls properties of existing objects.

System Variables

DIMCLRT

Assigns colors to dimension text.

DIMDSEP

Specifies a single-character decimal separator to use when creating dimensions whose unit format is decimal.

DIMJUST

Controls the horizontal positioning of dimension text.

DIMTAD

Controls the vertical position of text in relation to the dimension line.

DIMTXTDIRECTION

Specifies the reading direction of the dimension text.

DIMTIH

Controls the position of dimension text inside the extension lines for all dimension types except Ordinate.

DIMTMOVE

Sets dimension text movement rules.

DIMTOH

Controls the position of dimension text outside the extension lines.

DIMTVP

Controls the vertical position of dimension text above or below the dimension line.

DIMUPT

Controls options for user-positioned text.

MTEXTED

Sets the application for editing multiline text objects.

Utilities

No entries

Command Modifiers

No entries

Modify Dimension Geometry

Grip editing is the quickest and easiest way to modify the location of dimension elements. How you edit dimensions depends whether the dimension is associative.

You can modify dimensions with the editing commands and with grip editing. Grip editing is the quickest and easiest way to modify dimensions. How you edit dimensions depends on whether the dimension is associative.

Modify Associative Dimensions

Associative dimensions retain their associativity to dimensioned objects through many editing commands if both the dimension and the associated geometry are selected and operated on with a single command. For example, if a dimension and its associated geometry are moved, copied, or arrayed in the same command, each dimension retains associativity with its respective geometry.

In some circumstances, dimensions are automatically disassociated, including

- If the associated geometric object is erased

- If the associated geometric object undergoes a Boolean operation such as UNION or SUBTRACT
- If grip editing is used to stretch a dimension parallel to its dimension line
- If the association to a geometric object is specified using the Apparent Intersection object snap, and the geometric object is moved so that the apparent intersection no longer exists

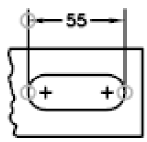
In other circumstances, a dimension may become partially associated. For example, if a linear dimension is associated with the endpoints of two geometric objects and one of the objects is erased, the remaining association is preserved. The disassociated end of the linear dimension may then be associated with another geometric object using DIMREASSOCIATE.

NOTE The command prompt displays a warning message if a dimension is disassociated.

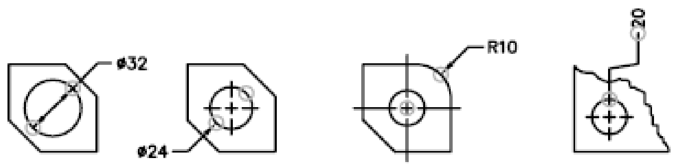
Modify Nonassociative Dimensions

For nonassociative dimensions, when you edit dimensioned objects, you must include the relevant dimension definition points in the selection set, or the dimension is not updated. Definition points determine the dimension location. For example, to stretch a dimension, you must include the appropriate definition points in the selection set. You can easily include them by turning on grips and selecting the object so that the grips are highlighted.

The definition points for each type of dimension are indicated in the following illustrations. The middle point of the dimension text is a definition point for all dimension types.



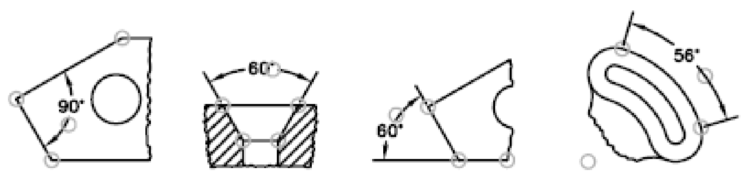
Linear: extension line origins and intersection of first extension line and dimension line



Diameter: selection point and opposite point

Radius: selection point and center

ordinate: feature location and leader endpoint



Three-Point Angular: angle vertex, extension line origins, and dimension line arc

Two-Point Angular: extension line origins, and dimension line arc

If no angle vertex is shown, definition points are placed at the ends of the lines that form the angle. In the two-line angular example, a definition point is placed at the center point of the dimensioned arc.

NOTE Definition points are drawn on a special layer named DEFPOINTS, which is not plotted.

Modify Exploded Dimensions

You can edit exploded dimensions as you would any other objects because an exploded dimension is a collection of separate objects: lines, 2D solids, and text. Occasionally you may need to explode a dimension to make changes such as creating a break in a dimension line or extension line. Once a dimension is exploded, you cannot reassociate the dimension into a dimension object.

See also:

- Control Dimension Geometry on page 1635

Quick Reference

Commands

DIMEDIT

Edits dimension text and extension lines.

DIMDISASSOCIATE

Removes associativity from selected dimensions.

DIMREASSOCIATE

Associates or reassociates selected dimensions to objects or points on objects.

EXPLODE

Breaks a compound object into its component objects.

STRETCH

Stretches objects crossed by a selection window or polygon.

System Variables

DIMASSOC

Controls the associativity of dimension objects and whether dimensions are exploded.

Utilities

No entries

Command Modifiers

No entries

Change Dimension Associativity

You may need to change the associativity of dimensions in several circumstances including adding associativity to dimensions created in previous releases.

You may need to change the associativity of dimensions in several circumstances such as the following:

- Redefine the associativity of dimensions in drawings that have been edited significantly.
- Add associativity to dimensions that have been partially disassociated.
- Add associativity to dimensions in legacy drawings.
- Remove associativity from dimensions in drawings that will be used by people working in releases prior to AutoCAD 2002, but who do not want any proxy objects in the drawings.

Reassociate Dimensions to Different Objects

With DIMREASSOCIATE, you can select one or more dimensions and step through the extension-line origin points of each dimension. For each extension-line origin point, you can specify a new *association point* on a geometric object. Association points determine the attachment of extension lines to locations on geometric objects.

NOTE When you create or modify associative dimensions, it is important to locate their association points carefully so that if you make a future design change, the geometric objects that you change will also change the dimensions associated with them.

When you use the DIMREASSOCIATE command, a marker is displayed that indicates whether each successive extension line origin point of the dimension is associative or nonassociative. A square with an X in it means that the point is associated with a location on an object, while an X without the square means that the point is not associated with an object. Use an object snap to specify the new association for the extension-line origin point or press ENTER to skip to the next extension-line origin point.

NOTE The marker disappears if you pan or zoom with a wheel mouse.

Change Nonassociative Dimensions to Associative

You can change all the nonassociative dimensions in a drawing to associative. Use QSELECT to select all nonassociative dimensions, and then use DIMREASSOCIATE to step through the dimensions, associating each one with locations on geometric objects.

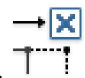
Change Associative Dimensions to Nonassociative

You can change all associative dimensions in a drawing to nonassociative dimensions. Use QSELECT to select all associative dimensions, and then use DIMDISASSOCIATE to convert them into nonassociative dimensions.

See also:

- Associative Dimensions on page 1627
- Save Drawings to Previous Drawing File Formats

To associate or reassociate a dimension

1. Click Annotate tab ➤ Dimensions panel ➤ Reassociate. 
2. Select one or more dimensions to associate or reassociate.
3. Do one of the following:
   - Specify the new location of the extension-line origin point.
   - Enter **s** and select a geometric object to associate with the dimension.
   - Press ENTER to skip to the next extension-line origin point.
   - Press ESC to end the command but keep any associations you made up to that point.
4. Repeat the previous step as needed.

Command entry: DIMREASSOCIATE

To disassociate a dimension

1. At the command prompt, enter DIMDISASSOCIATE.
2. Select one or more dimensions to disassociate and press ENTER when you finish.

Quick Reference

Commands

DIMDISASSOCIATE

Removes associativity from selected dimensions.

DIMREASSOCIATE

Associates or reassociates selected dimensions to objects or points on objects.

DIMREGEN

Updates the locations of all associative dimensions.

EXPLODE

Breaks a compound object into its component objects.

System Variables

DIMASSOC

Controls the associativity of dimension objects and whether dimensions are exploded.

Utilities

No entries

Command Modifiers

No entries

Add Geometric Tolerances

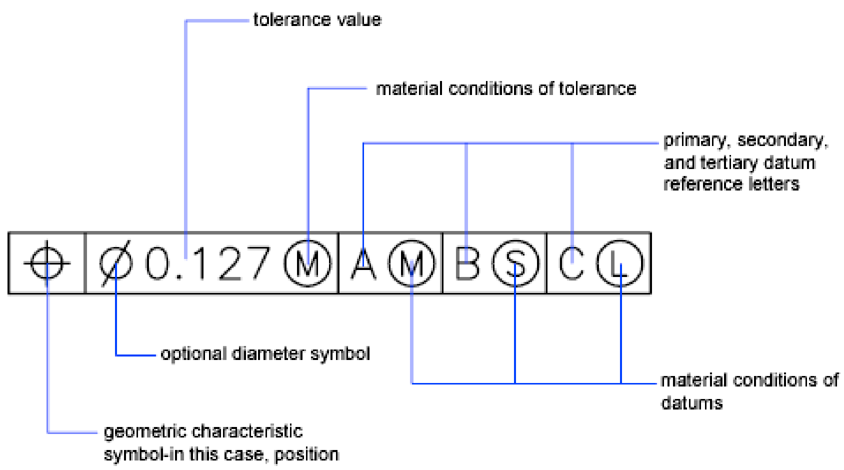
You can add geometric tolerances that show acceptable deviations of form, profile, orientation, location, and runout of a feature.

Overview of Geometric Tolerances

Geometric tolerances show acceptable deviations of form, profile, orientation, location, and runout of a feature.

You add geometric tolerances in feature control frames. These frames contain all the tolerance information for a single dimension. Geometric tolerances can be created with or without leader lines, depending on whether you create them with TOLERANCE or LEADER.

A feature control frame consists of two or more components. The first feature control frame contains a symbol that represents the geometric characteristic to which a tolerance is being applied, for example, location, profile, form, orientation, or runout. Form tolerances control straightness, flatness, circularity and cylindricity; profiles control line and surface. In the illustration, the characteristic is position.



You can use most editing commands to change feature control frames, and you can snap to them using the object snap modes. You can also edit them with grips.

NOTE Unlike dimensions and leaders, geometric tolerances cannot be associated with geometric objects.

You can also create tolerances. For more information about creating and working with an annotative tolerances, see Create Annotative Dimensions and Tolerances on page 1409.

See also:

- Scale Annotations on page 1393

To create geometric tolerances

1 Click Annotate tab ➤ Dimensions panel ➤ Tolerance. 

2 In the Geometric Tolerance dialog box, click the first square under Sym and select a symbol to insert.

3 Under Tolerance 1, click the first black box to insert a diameter symbol.

4 In the Text box, enter the first tolerance value.

5 To add a material condition (optional), click the second black box and click a symbol in the Material Conditions dialog box to insert it.

6 In the Geometric Tolerance dialog box, add a second tolerance value (optional) in the same way as the first tolerance value.

7 Under Datum 1, Datum 2, Datum 3, enter the datum reference letter.

8 Click the black box to insert a material condition symbol for each datum reference.

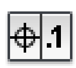
9 In the Height box, enter a height.

10 Click the Projected Tolerance Zone box to insert the symbol.

11 In the Datum Identifier box, add a datum value.

12 Click OK.

13 In the drawing, specify a location for the feature control frame.

Toolbar: Dimension 
Command entry: TOLERANCE

To create a geometric tolerance with a leader

1 At the command prompt, enter **leader**.

2 Specify the start point of the leader.

3 Specify the second point of the leader.

4 Press ENTER twice to display the Annotation options.

5 Enter **t** (Tolerance), and create a feature control frame.

The feature control frame is attached to the endpoint of the leader.

Command entry: LEADER

Quick Reference

Commands

LEADER

Creates a line that connects annotation to a feature.

TOLERANCE

Creates geometric tolerances contained in a feature control frame.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Material Conditions

Material conditions apply to features that can vary in size.

The second compartment contains the tolerance value. Depending on the control type, the tolerance value is preceded by a diameter symbol and followed by a material condition symbol.

Material conditions apply to features that can vary in size:

- At *maximum material condition* (symbol M, also known as MMC), a feature contains the maximum amount of material stated in the limits.
- At MMC, a hole has minimum diameter, whereas a shaft has maximum diameter.
- At *least material condition* (symbol L, also known as LMC), a feature contains the minimum amount of material stated in the limits.

- At LMC, a hole has maximum diameter, whereas a shaft has minimum diameter.
- *Regardless of feature size* (symbol S, also known as RFS) means a feature can be any size within the stated limits.

Quick Reference

Commands

LEADER

Creates a line that connects annotation to a feature.

TOLERANCE

Creates geometric tolerances contained in a feature control frame.

System Variables

No entries

Utilities

No entries

Command Modifiers

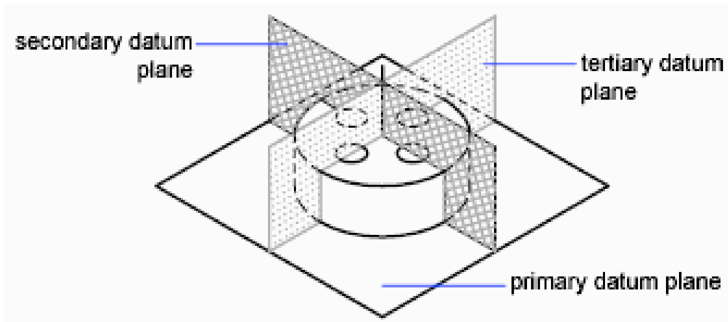
No entries

Datum Reference Frames

The tolerance values in the feature control frame are followed by up to three optional datum reference letters and their modifying symbols.

A datum is a theoretically exact point, axis, or plane from which you make measurements and verify dimensions. Usually, two or three mutually perpendicular planes perform this task best. These are jointly called the datum reference frame.

The following illustration shows a datum reference frame verifying the dimensions of the part.



Quick Reference

Commands

LEADER

Creates a line that connects annotation to a feature.

TOLERANCE

Creates geometric tolerances contained in a feature control frame.

System Variables

No entries

Utilities

No entries

Command Modifiers

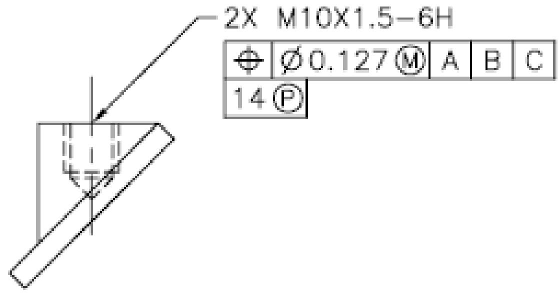
No entries

Projected Tolerance Zones

Projected tolerances are used to make the tolerance more specific.

Projected tolerances are specified in addition to positional tolerances to make the tolerance more specific. For example, projected tolerances control the perpendicularity tolerance zone of an embedded part.

The symbol for projected tolerance ( ) is preceded by a height value, which specifies the minimum projected tolerance zone. The projected tolerance zone height and symbol appear in a frame below the feature control frame, as shown in the following illustration.



Quick Reference

Commands

LEADER

Creates a line that connects annotation to a feature.

TOLERANCE

Creates geometric tolerances contained in a feature control frame.

System Variables

No entries

Utilities

No entries

Command Modifiers

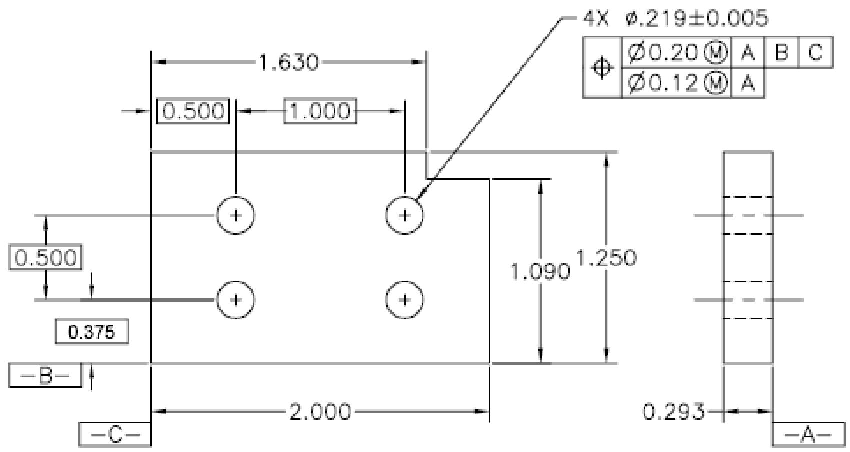
No entries

Composite Tolerances

A composite tolerance specifies two tolerances for the same geometric characteristic of a feature or for features that have different datum requirements. One tolerance relates to a pattern of features and the other tolerance to each feature within the pattern. The individual feature tolerance is more restrictive than the pattern tolerance.

In the following illustration, the point where datums A and B intersect is called the datum axis, the point from which the position of the pattern is calculated.

A composite tolerance could specify both the diameter of the pattern of holes and the diameter of each individual hole, as in the following illustration.



When you add composite tolerances to a drawing, you specify the first line of a feature control frame and then choose the same geometric characteristic symbol for the second line of the feature control frame. The geometric symbol compartment is extended over both lines. You can then create a second line of tolerance symbols.

Quick Reference

Commands

LEADER

Creates a line that connects annotation to a feature.

TOLERANCE

Creates geometric tolerances contained in a feature control frame.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Plot and Publish Drawings

Prepare Drawings for Plotting and Publishing

28

You prepare your drawing for plotting or publishing by specifying page setup settings. These settings are stored in the drawing file with the layout. Once a layout is established, you can modify the settings of its page setup or apply a different page setup.

Quick Start to Saving Settings for Plotting and Publishing

Preparing a drawing for plotting or publishing requires specifying many settings and options that define the output of your drawing. To save time, you can save these settings as a *named page setup.*

You can apply a named page setup to paper space layouts using the Page Setup Manager. You can also import a named page setup from another drawing and apply it to layouts in the current drawing.

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PSETUPIN

Imports a user-defined page setup into a new drawing layout.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Specify Page Setup Settings

Page setups are associated with layouts and stored in the drawing file. The settings specified in a page setup determine the appearance and format of your final output.

Overview of Page Setup Settings

A page setup is a collection of plot device and other settings that affect the appearance and format of your final output. These settings can be modified and applied to other layouts.

After you complete a drawing on the Model tab, you can begin creating a layout to plot by clicking a layout tab. When you click a layout tab for the first time, a single viewport is displayed on the page. A dashed line indicates the printable area of the paper for the currently configured paper size and plotter.

Once you have your layout set up, you specify the settings for the layout's page setup, which includes the plot device settings and other settings that affect the appearance and format of the output. The settings you specify in the page setup are stored in the drawing file with the layout. You can modify the settings of a page setup at any time.

By default, every initialized layout has a page setup associated with it. You can initialize a layout by clicking on its tab to activate the previously unused layout. A layout does not contain any plot settings before initialization. A layout must be initialized (its paper size can be defined in the page setup to any size other than 0 x 0) before it can be published. Once initialized, layouts can be drawn upon, published, and added to sheet sets as sheets (after the drawing has been saved). You can apply a named page setup saved with one layout to another layout. This creates a new page setup with the same settings as the first one.

If you want the Page Setup Manager to be displayed each time you begin a new drawing layout, select the Show Page Setup Manager for New Layouts option on the Display tab in the Options dialog box. If you don't want a viewport to be automatically created for each new layout, clear the Create Viewport in New Layouts option on the Display tab in the Options dialog box.

To modify the settings of a layout's page setup

1. Click the layout tab for which you want to modify the page setup settings.
2. Click Output tab ➤ Plot panel ➤ Page Setup Manager.
3. In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
4. Click Modify.
5. In the Page Setup dialog box, modify the required settings. Click OK.
6. In the Page Setup Manager, click Close.

Toolbar: Layouts
Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

To apply a layout's named page setup to another layout

1. Click the layout tab to which you want to apply another layout's page setup settings.
2. Click Output tab ➤ Plot panel ➤ Page Setup Manager.
3. In the Page Setup Manager, Page Setups area, select a named page setup that you want to apply to the layout selected in the drawing area.
4. Click Set Current.
5. Click Close.

Toolbar: Layouts 
Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Select a Printer or Plotter for a Layout

When you create a layout, you must select a printing or plotting device in the Page Setup dialog box in order for the layout to be printed or plotted. Once you've selected a device, you can view details about the name and location of the device, and you can change the device's configuration.

The printer or plotter you select in the Page Setup dialog box determines the printable area of the layout. This printable area is indicated by the dashed line in the layout. If you change the paper size or the printing or plotting device, it may change the printable area of your drawing page.

See also:

- Select a Printer or Plotter on page 1791
- "Control PC3 File Device and Document Settings" in the *Driver and Peripheral Guide*

To select a printer or plotter for a layout

1. Click the layout tab for which you want to specify a printer or plotter.
2. Click Output tab ➤ Plot panel ➤ Page Setup Manager. 
3. In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
4. Click Modify.
5. In the Page Setup dialog box, under Printer/Plotter, select a printer or plotter from the list. Click OK.
6. In the Page Setup Manager, click Close.

Toolbar: Layouts 
Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab and select Page Setup Manager.

To change the configuration of a printer or plotter specified in a page setup

1. Click the layout tab for which you want to specify a printer or plotter.
2. Click Output tab ➤ Plot panel ➤ Page Setup Manager. 
3. In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
4. Click Modify.
5. In the Page Setup dialog box, under Printer/Plotter, click Properties.
6. In the Plotter Configuration Editor, change the required settings. Click OK.
7. In the Page Setup dialog box, click OK.
8. In the Page Setup Manager, click Close.

Toolbar: Layouts 

Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PLOT

Plots a drawing to a plotter, printer, or file.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Select a Paper Size for a Layout

You can select a paper size from a standard list, or you can add custom paper sizes using the Plotter Configuration Editor.

You can select a paper size from a standard list. The paper sizes available in the list are determined by the plot device that is currently selected for the layout. If your plotter is configured for raster output, you must specify the output size in pixels. You can add custom paper sizes, which are stored in the plotter configuration (PC3) file, using the Plotter Configuration Editor.

If you are using a system printer, the paper size is determined by the document defaults that are set in the Windows Control Panel. The default paper size is displayed in the Page Setup dialog box when you create a new layout for that configured device. If you change the paper size in the Page Setup dialog box, the new paper size is saved with the layout and overrides the size saved in the plotter configuration (PC3) file.

See also:

- "Control PC3 File Device and Document Settings" in the *Driver and Peripheral Guide*

To set the paper size for a layout

1. Click the layout tab for which you want to set the paper size.
2. Click Output tab ➤ Plot panel ➤ Page Setup Manager. 
3. In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
4. Click Modify.
5. In the Page Setup dialog box, under Paper Size, select a paper size from the list. Click OK.
6. In the Page Setup Manager, click Close.

Toolbar: Layouts 
Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

To start the Plotter Configuration Editor

1. Click Output tab ➤ Plot panel ➤ Plotter Manager. 
2. In the Plotter Manager, double-click the plotter configuration (PC3) file you want to edit.

   The Plotter Configuration Editor is displayed.

Command entry: PLOTTERMANAGER

To add a custom paper size from scratch

1. Click Output tab ➤ Plot panel ➤ Plotter Manager. 

2 In the Plotter Manager, double-click the plotter configuration (PC3) file you want to edit.

3 In the Plotter Configuration Editor, Device and Document Settings tab, double-click User-Defined Paper Sizes & Calibration to display the calibration and paper size options.

4 Select Custom Paper Sizes.

5 Under Custom Paper Sizes, click Add.

6 In the Custom Paper Size wizard, Begin page, select Start from Scratch. Click Next.

7 On the Media Bounds page, in the Units list, select either Inches or Millimeters for paper size.

   When a nondimensional raster image, such as BMP or TIFF, is plotted, the size of the plot is specified in pixels, not inches or millimeters.

8 In the Width and Length lists, specify the paper width and length. Click Next.

   NOTE Each plotter has a maximum printable area determined by where it grips the paper and how far the pen shuttle reaches. If you are creating a paper size that is larger than the paper sizes offered in the Custom Paper Size wizard, verify that the plotter is capable of plotting the new dimensions.

   ---

9 On the Printable Area page, use Top, Bottom, Left, and Right to specify the printable area. Click Next.

10 On the Paper Size Name page, enter a name for the paper size. Click Next.

11 On the File Name page, enter a name for the PMP file.

12 On the Finish page, specify whether the paper source is Sheet-Fed or Roll-Fed.

13 Click Print Test Page to verify the custom size.

   A cross is printed that defines the paper size and a rectangle that defines the printable area. If all four sides of the rectangle are not printed, increase the printable area.

14 Click Finish to exit the Custom Paper Size wizard.

Command entry: PLOTTERMANAGER

To add a new custom paper size starting from an existing paper size

1 Click Output tab ➤ Plot panel ➤ Plotter Manager.

2 In the Plotter Manager, double-click the plotter configuration (PC3) file you want to edit.

3 In the Plotter Configuration Editor, Device and Document Settings tab, double-click User-Defined Paper Sizes & Calibration to display the calibration and paper size settings.

4 Select Custom Paper Sizes.

5 Under Custom Paper Sizes, click Add.

6 In the Custom Paper Size wizard, Begin page, select Use Existing.

7 In the list of existing standard paper sizes, select a paper size on which to base the custom paper size you are creating.

8 Follow the instructions in To add a custom paper size from scratch on page 1753 to continue through the Custom Paper Size wizard.

The new paper size is a user-defined size, not a standard size.

9 Click Finish to exit the Custom Paper Size wizard.

Command entry: PLOTTERMANAGER

To edit a custom paper size

1 Click Output tab ➤ Plot panel ➤ Plotter Manager.

2 In the Plotter Manager, double-click the plotter configuration (PC3) file you want to edit.

3 In the Plotter Configuration Editor, Device and Document Settings tab, double-click User-Defined Paper Sizes & Calibration to display the calibration and paper size settings.

4 Select Custom Paper Sizes.

5 Under Custom Paper Sizes, select a paper size from the list. Click Edit.

6 In the Custom Paper Size wizard, make changes to the paper size, printable area, custom paper size name, and source.

7 Click Finish to exit the Custom Paper Size wizard.

Command entry: PLOTTERMANAGER

To delete a custom paper size

1 Click Output tab ➤ Plot panel ➤ Plotter Manager. 

2 In the Plotter Manager, double-click the plotter configuration (PC3) file you want to edit.

3 In the Plotter Configuration Editor, Device and Document Settings tab, double-click User-Defined Paper Sizes & Calibration to display the calibration and paper size settings.

4 Click Custom Paper Sizes.

5 Under Custom Paper Sizes, select a paper size from the list.

6 Click Delete.

Command entry: PLOTTERMANAGER

To modify a standard paper size

1 Click Output tab ➤ Plot panel ➤ Plotter Manager. 

2 In the Plotter Manager, double-click the Plotter Configuration (PC3) file you want to edit.

3 In the Plotter Configuration Editor, Device and Document Settings tab, double-click User-Defined Paper Sizes & Calibration to display the calibration and paper size settings.

4 Select Modify Standard Paper Sizes.

5 Under Modify Standard Paper Sizes, select the paper size you want to adjust. Click Modify.

6 In the Custom Paper Size wizard, adjust the printable area as necessary. Click Finish to exit the Custom Paper Size wizard.

Command entry: PLOTTERMANAGER

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PLOTTERMANAGER

Displays the Plotter Manager, where you can add or edit a plotter configuration.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Set the Plot Area of a Layout

You can specify the plot area to determine what will be included in the plot.

When you prepare to plot from the Model tab or a layout tab, you can specify the plot area to determine what will be included in the plot. When you create a new layout, the default Plot Area option is Layout. Layout plots all objects within the printable area of the specified paper size.

The Display Plot Area option plots all the objects displayed in the drawing. The Extents Plot Area option plots all the visible objects in the drawing. The View Plot Area option plots a saved view. You can use the Window Plot Area option to define an area to be plotted.

See also:

- Specify the Area to Plot on page 1793

To set the plot area and adjust the display

1 Click the layout tab for which you want to set the plot area and adjust the display.

2 Click Output tab ➤ Plot panel ➤ Page Setup Manager.

3 In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.

4 Click Modify.

5 In the Page Setup dialog box, under Plot Area, select one of the following options:

   - **Layout.** Prints all objects within the printable area of the paper. This option is only available from a layout tab.
   - **Limits.** Prints or plots the current grid limits. This option is only available from the Model tab.
   - **Extents.** Plots all objects in the drawing.
   - **Display.** Plots all objects displayed in the drawing area.
   - **View.** Prints or plots a saved view. Select a named view from the list provided.
   - **Window.** Plots objects in the area you define. Select the Window option, and then respond to the prompts to define the area. Click the Window button to edit the defined area.

6 Click OK.

7 In the Page Setup Manager, click Close.

Toolbar: Layouts
Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

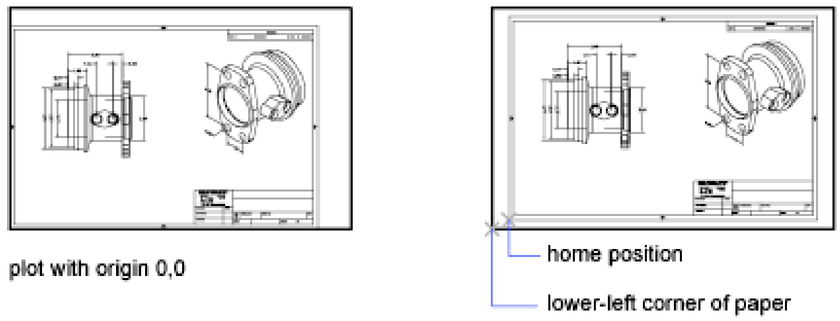
Adjust the Plot Offset of a Layout

The printable area of a drawing sheet is defined by the selected output device and is represented by the dashed line in a layout. When you change the output device, the printable area may change.

The plot offset specifies an offset of the plot area relative to the lower-left corner (the origin) of the printable area or the edge of the paper, depending on the Specify Plot Offset Relative To option specified in the Options dialog box, Plot and Publish tab. The Plot Offset area of the Plot dialog box displays the specified plot offset option in parentheses.

You can offset the geometry on the paper by entering a positive or negative value in the X and Y Offset boxes. However, this may result in the plot area being clipped.

If you choose to plot an area other than the entire layout, you can also center the plot on the sheet of paper.



plot with origin 0,0

plot with origin -1.0,-0.5

To adjust the plot offset of a layout

1. Click the layout tab for which you want to adjust the plot offset.
2. Click Output tab ➤ Plot panel ➤ Page Setup Manager. 
3. In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
4. Click Modify.
5. In the Page Setup dialog box, under Plot Offset, enter a value in units for *X* or *Y* or both. Click OK.
6. In the Page Setup Manager, click Close.

Toolbar: Layouts 
Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

System Variables

PLOTOFFSET

Controls whether the plot offset is relative to the printable area or to the edge of the paper.

Utilities

No entries

Command Modifiers

No entries

Set the Plot Scale for a Layout

When you plot a drawing layout, you can either specify a precise scale for the layout or fit the image to the paper.

Normally, you plot a layout at a 1:1 scale. To specify a different scale for the layout, set the plot scale for the layout in the Page Setup or the Plot dialog box. In those dialog boxes, you can select a scale from a list or enter a scale.

NOTE You can modify the list of scales with SCALELISTEDIT.

When you are reviewing an early draft view, a precise scale is not always important. You can use the Fit to Paper setting to plot the layout at the largest possible size that fits the paper.

See also:

■ Scale Views in Layout Viewports on page 456

■ Draw, Scale, and Annotate in Model Space on page 425

To set the plot scale in a layout

1 Click the layout tab for which you want to set the plot scale.

2 Click Output tab ➤ Plot panel ➤ Page Setup Manager. 

3 In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.

4 Click Modify.

5 In the Page Setup dialog box, under Plot Scale, select a scale from the Scale list.

  The default scale when plotting a layout is 1:1. To set a custom plot scale, enter values in the Inches or Millimeters box and the Units box. The type of unit is determined by the paper size, but you can change it in the list box.

6 Click OK.

7 In the Page Setup Manager, click Close.

Toolbar: Layouts
Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab. Click Page Setup.

To set the Fit to Paper option when plotting

1 Click the layout tab for which you want to set the plot scale to Fit to Paper.

2 Click Output tab ➤ Plot panel ➤ Page Setup Manager.

3 In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.

4 Click Modify.

5 In the Page Setup dialog box, Under Plot Scale, select Fit to Paper.

  NOTE If the Plot Area is set to Layout, you cannot select the Fit to Paper option.

6 Click OK.

7 In the Page Setup Manager, click Close.

Toolbar: Layouts
Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

Quick Reference

Commands

SCALELISTEDIT

Controls the list of scales available for layout viewports, page layouts, and plotting.

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Set the Lineweight Scale for a Layout

You can scale lineweights proportionately in a layout with the plot scale.

Typically, lineweights specify the line width of plotted objects and are plotted with the line width size regardless of the plot scale. Most often, you use the default plot scale of 1:1 when plotting a layout. However, if you want to plot an E-size layout that is scaled to fit on an A-size sheet of paper, for example, you can specify lineweights to be scaled in proportion to the new plot scale.

See also:

- Control Lineweights on page 614

To scale lineweights in a layout

1 Click the layout tab for which you want to scale lineweights.

2 Click Output tab ➤ Plot panel ➤ Page Setup Manager. 

3 In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.

4 Click Modify.

5 In the Page Setup dialog box, under Plot Scale, select Scale Lineweights.

   The lineweights in the current layout are scaled in proportion to the designated plot scale. When you are working in the Model tab, this option is not available.

6 Click OK.

7 In the Page Setup Manager, click Close.

Toolbar: Layouts 
Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Select a Plot Style Table for a Layout

A plot style table is a collection of plot styles assigned to a layout or the Model tab. A plot style is an object property, similar to linetype and color. A plot style can be assigned to an object or assigned to a layer. A plot style controls an object's plotted properties.

You can also create a new plot style table to save in the page setup for the layout or edit an existing plot style table.

If you select the Display Plot Styles option under Plot Style Table (Pen Assignments), the properties of the plot styles assigned to objects are displayed in the selected layout.

See also:

- Control How Objects Are Plotted on page 1801

To select a plot style table for a layout

1 Click the layout tab for which you want to select a plot style table.

2 Click Output tab ➤ Plot panel ➤ Page Setup Manager. 

3 In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.

4 Click Modify.

5 In the Page Setup dialog box, Under Plot Style Table, select a Plot Style Table from the list.

6 Click OK.

7 In the Page Setup Manager, click Close.

Toolbar: Layouts 
Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

To create a new plot style table for a layout

1 Click the layout tab for which you want to create a new plot style table.

2 Click Output tab ➤ Plot panel ➤ Page Setup Manager. 

3 In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.

4 Click Modify.

5 In the Page Setup dialog box, Under Plot Style Table, select New from the list.

6 Follow the instructions in the wizard that is shown (the Add Color-Dependent Plot Style Table wizard or the Add Named Plot Style Table wizard).

7 In the Page Setup dialog box, click OK.

8 In the Page Setup Manager, click Close.

Toolbar: Layouts 
Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

To edit a plot style table for a layout

1 Click the layout tab for which you want to edit a plot style table.

2 Click Output tab ➤ Plot panel ➤ Page Setup Manager. 

3 In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.

4 Click Modify.

5 In the Page Setup dialog box, under Plot Style Table, select the plot style table you want to edit from the list.

6 Click the Edit button.

7 In the Plot Style Table Editor, make the required changes. Click Save & Close.

8 In the Page Setup dialog box, click OK.

9 In the Page Setup Manager, click Close.

Toolbar: Layouts
Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

To display plot styles in a layout

1. Click the layout tab for which you want to display plot styles.
2. Click Output tab ➤ Plot panel ➤ Page Setup Manager.
3. In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
4. Click Modify.
5. In the Page Setup dialog box, under Plot Style Table, select the Display Plot Styles option.
6. Click OK.
7. In the Page Setup Manager, click Close.

Toolbar: Layouts
Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

STYLESMANAGER

Displays the Plot Style Manager, where you can revise plot style tables.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Set Shaded Viewport and Plot Options for a Layout

Shaded viewport and plot options settings affect how objects are plotted and are saved in the page setup.

Shaded viewport and plot options affect how objects are plotted. The options for shaded viewport plotting give you a large degree of flexibility in conveying your three-dimensional designs to others. You can convey your design intent by choosing how viewports are plotted and by specifying resolution levels.

Shaded Viewport Plotting Options

With shaded plotting options, you can choose whether to plot a set of shaded objects using the As Displayed, Wireframe, Hidden, or Rendered option.

Shaded viewport plotting options apply to all objects in viewports and model space. If you use the Shaded or Rendered options, plot style tables included in the page setup do not affect plots. If you use the Render option, two-dimensional wireframe objects, such as lines, arcs, and text, are not plotted.

NOTE Shaded viewport plotting requires a raster-capable device. Most modern plotters and printers are raster-capable devices.

Plot Options

The following options that can be specified for layouts affect how objects are plotted.

- **Plot Object Lineweights.** Specifies that lineweights assigned to objects and layers are plotted.
- **Plot with Plot Styles.** Specifies that the drawing is plotted using plot styles. Selecting this option automatically plots lineweights. If you do not select

this option, objects are plotted with their assigned properties and not with the plot style overrides.

- **Plot Paper Space Last.** Specifies that objects in model space are plotted before those in paper space.
- **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.

See also:

- Set Shaded Viewport Options on page 1805
- Set Options for Plotted Objects on page 1809

To set shaded viewport options for a layout

1. Click the layout tab for which you want to set shaded viewport options.
2. Click Output tab ➤ Plot panel ➤ Page Setup Manager. 
3. In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
4. Click Modify.
5. In the Page Setup dialog box, under Shaded Viewport Options, select the required settings.
6. Click OK.
7. In the Page Setup Manager, click Close.

Toolbar: Layouts 
Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

To set plot options for a layout

1. Click the layout tab for which you want to set plot options.

2 Click Output tab ➤ Plot panel ➤ Page Setup Manager. 

3 In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.

4 Click Modify.

5 In the Page Setup dialog box, under Plot Options, select the required settings.

6 Click OK.

7 In the Page Setup Manager, click Close.

Toolbar: Layouts 
Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Determine the Drawing Orientation of a Layout

You can specify the orientation of the drawing on the paper using the Landscape and Portrait settings. Landscape orients the drawing on the paper so that the long edge of the paper is horizontal, and Portrait orients the paper so that the short edge is horizontal. Changing the orientation creates the effect of rotating the paper underneath the drawing.

In either landscape or portrait orientation, you can select Plot Upside-Down to control whether the top or bottom of the drawing is plotted first.

Although you can specify the drawing orientation in both the Page Setup dialog box and the Plot dialog box, the Page Setup settings are always saved and reflected in the layout. In the Plot dialog box, you can override the page setup settings for a single plot; however, the settings you apply are not saved in the layout. To save the settings you apply using the Plot dialog box, click the Apply to Layout button in the Plot dialog box.

If you change the drawing orientation, the layout origin remains in the lower-left corner of the rotated page.

To set the orientation of the plotted drawing

1. Click the layout tab for which you want to set the drawing orientation.
2. Click Output tab ➤ Plot panel ➤ Page Setup Manager. 
3. In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
4. Click Modify.

5 In the Page Setup dialog box, under Drawing Orientation, do one of the following:
   - If your drawing is horizontal, select Landscape.
   - If your drawing is vertical, select Portrait.
   - To rotate 180 degrees, select either Portrait or Landscape, and then select Plot Upside-Down.

6 Click OK.

7 In the Page Setup Manager, click Close.

Toolbar: Layouts 
Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Use the Layout Wizard to Specify Layout Settings

You can create a new layout using the Create Layout wizard.

The wizard prompts you for information about the layout settings, including

- A name for the new layout
- The printer associated with the layout
- A paper size to use for the layout
- The orientation of the drawing on the paper
- A title block
- Viewport setup information
- A location for the viewport configuration in the layout

You can edit the information entered in the wizard later. Click Output tab ➤ Plot panel ➤ Page Setup Manger.  In the Page Setup Manager, click Modify.

To create a layout using the wizard

1. Click Insert menu ➤ Layout ➤ Layout Wizard.
2. On each page of the Create Layout wizard, select the appropriate settings for the new layout.

   When finished, the new layout will be the current layout tab.

Command entry: LAYOUTWIZARD

Quick Reference

Commands

LAYOUTWIZARD

Creates a new layout tab and specifies page and plot settings.

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Import PCP or PC2 Settings into a Layout

You can import layout and plot settings contained in PCP or PC2 files into the current layout.

If you work with drawings created in AutoCAD Release 14 or earlier, you can choose to import layout and plot settings contained in a PCP or PC2 file and apply them to the current layout. Settings that are saved in a PCP or PC2 file include

- Plot Area
- Rotation
- Paper Size
- Plot Scale
- Plot Origin
- Plot Offset

In addition, a PC2 file contains any resolution information that has been modified by a plotter calibration. Pen assignment information can also be imported and saved in a plot style table using the Add Plot Style Table wizard.

To import plotting device and pen settings information, you can use the Import PCP or PC2 Plot Settings wizard to choose a PCP or PC2 file whose settings you want to import. You can also choose to modify any of the imported settings using the Page Setup dialog box.

To import PCP or PC2 settings into the current layout

1 At the command prompt, enter **pcinwizard**.

2 In the Import PCP or PC2 Plot Settings wizard, select the PCP or PC2 file whose settings you want to import into the current layout.

Quick Reference

Commands

PCINWIZARD

Displays a wizard to import PCP and PC2 configuration file plot settings into the Model tab or current layout.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Create and Use Named Page Setups

You can save plot device and other page setup settings as named page setups that can be modified and imported into other drawings.

You can create named page setups and apply them to other layouts in your drawing. Named page setups are saved in the drawing file and can be imported into other drawing files and applied to other layouts. You can also modify the settings of named page setups. If you modify a named page setup, you can choose whether the modifications apply to the current layout or to all the layouts in the current drawing that use the named page setup.

If you want to plot the same layout more than one way, or if you want to specify the same output options for several layouts, use named page setups.

You can apply different named page setups to the same layout to achieve specific results when plotting. For example, you might create the named page setups in the following table to control scaling and paper size.

Page setup name	Description
NoScaling	Plot at scale 1:1, E-size sheet
Scale 1 to 2	Plot at scale 1:2, C-size sheet
Draft	Plot to the draft-quality plotter
Final	Plot to the high-quality plotter
Fit-to-Paper	Fit to Paper, A-size sheet

Once you specify a named page setup for a layout, whenever you plot or publish the layout, it is plotted or published with the settings specified in the named page setup set for the layout.

To create a new named page setup

1. Click Output tab ➤ Plot panel ➤ Page Setup Manager. 
2. In the Page Setup Manager, Page Setups area, click New.
3. In the New Page Setup dialog box, enter a name for the new page setup.
4. Under Start With, select a page setup in the list. The settings specified in the selected page setup will be displayed in the Page Setup dialog box after you click OK.
5. Click OK.
6. In the Page Setup dialog box, change any required settings. Click OK.

   The new page setup is displayed in the Page Setups list in the Page Setup Manager.
7. To apply the new page setup to the current layout, in the Page Setup Manager, click Set Current.
8. In the Page Setup Manager, click Close.

Toolbar: Layouts
Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

To apply a named page setup to a layout

1 Click the layout tab for which you want to apply a named page setup.

2 Click Output tab ➤ Plot panel ➤ Page Setup Manager.

3 In the Page Setup Manager, under Page Setups, select a named page setup from the list.

NOTE A page setup that is not named is shown in the Page Setup Manager with an asterisk on either side of its layout name. An unnamed page setup does not have the same functionality as a named page setup. For example, if you set an unnamed page setup as the current page setup for another layout, the values are applied to the layout. If you make changes to the original unnamed page setup, those changes do not carry over to the layout to which the unnamed page setup was applied.

---

4 Click Set Current.

5 Click Close.

Toolbar: Layouts
Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

To modify a named page setup

1 Click Output tab ➤ Plot panel ➤ Page Setup Manager.

2 In the Page Setup Manager, under Page Setups, select a named page setup from the list.

NOTE A page setup that is not named is shown in the Page Setup Manager with an asterisk on either side of its layout name. An unnamed page setup does not have the same functionality as a named page setup. For example, if you set an unnamed page setup as the current page setup for another layout, the values are applied to the layout. If you make changes to the original unnamed page setup, those changes do not carry over to the layout to which the unnamed page setup was applied.

3 Click Modify.

4 In the Page Setup dialog box, make the required changes. Click OK.

5 In the Page Setup Manager, click Close.

Toolbar: Layouts 
Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

To import named page setups from another drawing

1 Click Output tab ➤ Plot panel ➤ Page Setup Manager. 

2 In the Page Setup Manager, click Import.

3 In the Select Page Setup From File dialog box, select a drawing file from which you want to import one or more named page setups. Click Import.

4 In the Import Page Setups dialog box, select one or more page setups to import. Click OK.

If a page setup with the same name already exists in the drawing, you can redefine the settings of the existing one with the settings of the imported page setup, or you can cancel the operation.

The imported page setups are displayed in the Page Setup Manager in the list of page setups.

NOTE You can import both model space and layout page setups at the same time. However, an imported model space page setup is shown in the Page Setup Manager only if the model tab was current when you opened the Page Setup Manager. Similarly, an imported layout page setup is shown in the Page Setup Manager only if a layout tab was current when you opened the Page Setup Manager.

5 In the Page Setup Manager, click Close.

Toolbar: Layouts
Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

To delete a named page setup

1 Click Output tab ➤ Plot panel ➤ Page Setup Manager.

2 In the Page Setup Manager, under Page Setups, right-click the named page setup you want to delete. Click Delete.

NOTE A page setup that is not named is shown in the Page Setup Manager with an asterisk on either side of its layout name. An unnamed page setup does not have the same functionality as a named page setup. For example, if you set an unnamed page setup as the current page setup for another layout, the values are applied to the layout. If you make changes to the original unnamed page setup, those changes do not carry over to the layout to which the unnamed page setup was applied.

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3 Click Close.

Toolbar: Layouts
Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

To rename a page setup

1 Click Output tab ➤ Plot panel ➤ Page Setup Manager.

2 In the Page Setup Manager, under Page Setups, right-click the named page setup you want to rename. Click Rename.

NOTE A page setup that is not named is shown in the Page Setup Manager with an asterisk on either side of its layout name. An unnamed page setup does not have the same functionality as a named page setup. For example, if you set an unnamed page setup as the current page setup for another layout, the values are applied to the layout. If you make changes to the original unnamed page setup, those changes do not carry over to the layout to which the unnamed page setup was applied.

3 Enter a new name for the page setup.

4 Click Close.

Toolbar: Layouts 
Command entry: PAGESETUP
Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Use Named Page Setups with Sheet Sets

You can use named page setups to specify the same output options for all the sheets in a sheet set.

You can create a sheet from scratch using the Sheet Set Manager, and then apply a named page setup to the layout.

You can also apply named page setups that are stored in the sheet set's page setup overrides DWT file to a single sheet or to an entire sheet set for a one-time publish operation.

See also:

- Create and Use Named Page Setups on page 1775

To apply a named page setup to a sheet created from scratch

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager. 

2 In the Open Sheet Set dialog box, select a sheet set. Click Open.

3 In the Sheet Set Manager, right click in the Sheets area. Click New Sheet.

4 In the New Sheet dialog box, enter a number and sheet title. Click OK.

5 In the Sheet Set Manager, under Sheets, double-click the new sheet.

6 Click Output tab ➤ Plot panel ➤ Page Setup Manager. 

7 In the Page Setup Manager, under Page Setups, select a named page setup. Click Set Current.

  Named page setups are not enclosed in asterisks.

8 Click Close.

Toolbar: Standard 
Command entry: SHEETSET

To apply a named page setup as an override when publishing sheets

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager. 

2 In the Open Sheet Set dialog box, select a sheet set. Click Open.

3 In the Sheet Set Manager, in the Sheets area, select a sheet set, a subset, or a sheet to publish.

4 At the top of the Sheet Set Manager, click the Publish button.

5 On the shortcut menu, place your cursor over Publish Using Page Setup Override.

6 On the submenu, click the named page setup you want to use.

Toolbar: Standard 
Command entry: SHEETSET

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PUBLISH

Publishes drawings to DWF, DWFx, and PDF files, or to plotters.

SHEETSET

Opens the Sheet Set Manager.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Plot Drawings

29

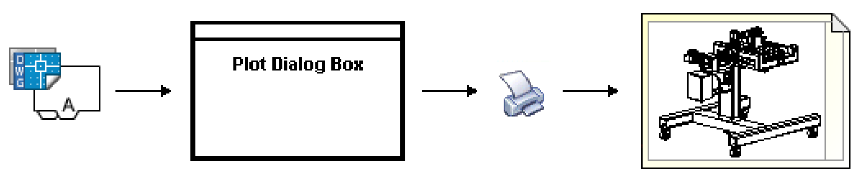
Once you have completed a drawing, you can use a number of methods to output the drawing. You can plot the drawing on paper or create a file for use with another application. In either case, you select the plot settings.

Quick Start to Plotting

To print a single layout or part of a drawing, use the Plot dialog box.

Use a named page setup or change the settings in the Plot dialog box to define the output of your drawing.

To output more than one drawing, use the Publish dialog box.



See also:

- "Quick Start to Publishing" on page 1

Quick Reference

Commands

PLOT

Plots a drawing to a plotter, printer, or file.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Overview of Plotting

Understanding terms and concepts that relate to plotting makes your first plotting experience in the program easier.

Plotter Manager

The Plotter Manager is a window that lists plotter configuration (PC3) files for every non-system printer that you install. Plotter configuration files can also be created for Windows® system printers if you want to use default properties different from those used by Windows. Plotter configuration settings specify port information, raster and vector graphics quality, paper sizes, and custom properties that depend on the plotter type.

The Plotter Manager contains the Add-a-Plotter wizard, which is the primary tool for creating plotter configurations. The Add-a-Plotter wizard prompts you for information about the plotter you want to set up.

Layouts

A layout represents a plotted page. You can create as many layouts as you need. Each layout is saved on its own layout tab and can be associated with a different page setup.

Elements that appear only on a plotted page, such as title blocks and notes, are drawn in paper space in a layout. The objects in the drawing are created in model space on the Model tab. To view these objects in the layout, you create layout viewports.

Layout Initialization

Layout initialization is a process in which a previously unused layout is made active, by clicking on its tab.

A layout does not contain any plot settings before initialization. Once initialized, layouts can be drawn upon, published, and added to sheet sets as sheets (after the drawing has been saved).

Page Setups

When you create a layout, you specify a plotter and settings such as page size and plot orientation. These settings are saved in a page setup. You can control these settings for layouts and for the Model tab using the Page Setup Manager. You can name and save page setups for use with other layouts.

If you don't specify all the settings in the Page Setup dialog box when you create a layout, you can set up the page just before you plot. Or you can override a page setup at plot time. You can use the new page setup temporarily for the current plot, or you can save the new page setup.

Plot Styles

A plot style controls how an object or layer is plotted by determining plotted properties such as lineweight, color, and fill style. Plot style tables collect groups of plot styles. The Plot Style Manager is a window that shows all the plot style tables available.

There are two plot style types: color-dependent and named. A drawing can use only one type of plot style table. You can convert a plot style table from one type to the other. You can also change the type of plot style table a drawing uses once it has been set.

For *color-dependent plot style tables,* an object's color determines how it is plotted. These plot style table files have *.ctb* extensions. You cannot assign color-dependent plot styles directly to objects. Instead, to control how an object is plotted, you change its color. For example, all objects assigned the color red in a drawing are plotted the same way.

Named plot style tables use plot styles that are assigned directly to objects and layers. These plot style table files have *.stb* extensions. Using them enables each object in a drawing to be plotted differently, independent of its color.

Plot Stamps

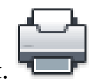
A plot stamp is a line of text that is added to your plot. You can specify where this text is located on the plot in the Plot Stamp dialog box. Turn this option on to add specified plot stamp information—including drawing name, layout name, date and time, and so on—to a drawing that is plotted to any device. You can choose to record the plot stamp information to a log file instead of plotting it, or in addition to plotting it.

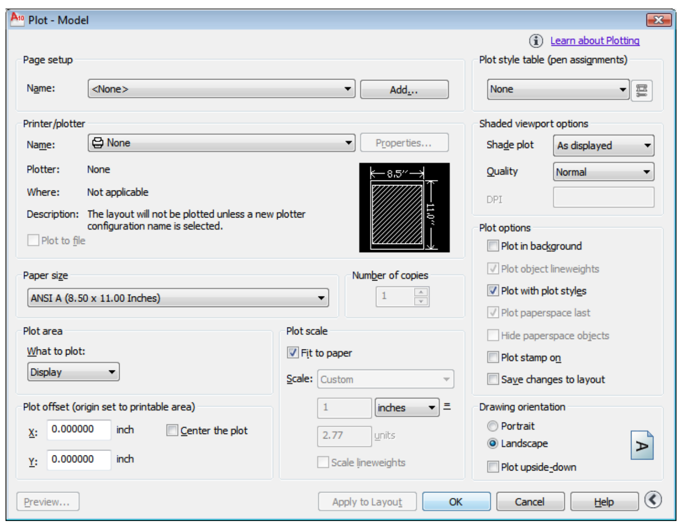
NOTE A drawing file or drawing template file that was created with an educational version will always be plotted with the following plot stamp: PRODUCED BY AN AUTODESK EDUCATIONAL PRODUCT. Blocks and xrefs created with an educational version and used in a commercial version will also result in the educational plot stamp being plotted.

See also:

- Create Multiple-View Drawing Layouts (Paper Space) on page 431
- "To view the custom properties Help" in the *Driver and Peripheral Guide*

To plot a drawing

1 Click Output tab ➤ Plot panel ➤ Plot. 



2 In the Plot dialog box, under Printer/Plotter, select a plotter from the Name list.

3 Under Paper Size, select a paper size from the Paper Size box.

4 (Optional) Under Number of Copies, enter the number of copies to plot.

5 Under Plot Area, specify the portion of your drawing to plot.

6 Under Plot Scale, select a scale from the Scale box.

7 For more options, click the More Options button.

8 (Optional) Under Plot Style Table (Pen Assignments), select a plot style table from the Name box.

9 (Optional) Under Shaded Viewport Options and Plot Options, select any appropriate settings.

NOTE Plot stamping happens at plot time and is not saved with the drawing.

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10 Under Drawing Orientation, select an orientation.

11 Click OK.

Toolbar: Standard 
Command entry: PLOT
Shortcut menu: Right-click the Model tab or a layout tab and click Plot.

To plot at the command prompt instead of in a dialog box, use -PLOT.

To turn background plotting on or off

1 Click Tools menu ➤ Options.

2 In the Options dialog box, Plot and Publish tab, under Background Processing Options, select or clear the Enable Background Plot When Plotting option.

3 Click OK.

NOTE When you plot in the background, you can return immediately to your drawing. While a plot job is being processed in the background, you can check the status of your job by placing your cursor over the plotter icon in the status tray. You can also view details about all completed jobs from the current program session.

Command entry: OPTIONS

To check the status of a plot job currently processing in the background

- In the status tray, place your mouse over the plotter icon. The tooltip displays the status of the plot job.

To cancel part or all of a plot job that is processing in the background

- In the status tray, right-click the plotter icon. Click Cancel Sheet <*sheetname*> or Cancel Entire Job.

To view details about jobs you have plotted

1. Do one of the following:
   - Click Output tab ➤ Plot panel ➤ View Details. 
   - In the status tray, click the plotter icon.
2. In the Plot and Publish Details dialog box, view details about plotted jobs.

Command entry: VIEWPLOTDETAILS
Shortcut menu: In the status tray, right-click the plotter icon. Click View Plot And Publish Details.

Quick Reference

Commands

OPTIONS

Customizes the program settings.

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PCINWIZARD

Displays a wizard to import PCP and PC2 configuration file plot settings into the Model tab or current layout.

PLOT

Plots a drawing to a plotter, printer, or file.

PLOTSTAMP

Places a plot stamp on a specified corner of each drawing and logs it to a file.

PLOTTERMANAGER

Displays the Plotter Manager, where you can add or edit a plotter configuration.

STYLESMANAGER

Displays the Plot Style Manager, where you can revise plot style tables.

VIEWPLOTDETAILS

Displays information about completed plot and publish jobs.

System Variables

BACKGROUNDPLOT

Controls whether background plotting is turned on or off for plotting and publishing.

Utilities

No entries

Command Modifiers

No entries

Use a Page Setup to Specify Plot Settings

You can use a page setup to specify the settings for your plot job. When you select a page setup in the Plot dialog box, the settings from the page setup are added to the Plot dialog box. You can choose to plot with those settings, or change any of the settings individually and then plot.

Any settings you specify in the Plot dialog box can be saved as a new named page setup by clicking the Add button in the Page Setup area.

Any settings specified in the Plot dialog box, whether you've applied a page setup from the Page Setup list, or changed the settings individually, can be saved to the layout for use the next time you plot.

To plot a drawing using the settings specified in a page setup

1 Click Output tab ➤ Plot panel ➤ Plot.

2 In the Plot dialog box, under Page Setups, select a page setup from the list.

3 (Optional) Change any of the individual settings in the Plot dialog box.

4 To save these settings to the layout, click Apply to Layout.

5 Click Plot.

Toolbar: Standard
Command entry: PLOT

To save plot settings as a new named page setup

1 Click Output tab ➤ Plot panel ➤ Plot.

2 In the Plot dialog box, change the required settings.

3 Under Page Setups, click Add.

4 In the Add Page Setup dialog box, enter a name for the new named page setup. Click OK.

Toolbar: Standard
Command entry: PLOT

To save plot settings to the layout

1 Select a layout tab.

2 Click Output tab ➤ Plot panel ➤ Plot.

3 In the Plot dialog box, under Page Setups, select a page setup or specify settings individually.

4 Click Apply to Layout.

Toolbar: Standard
Command entry: PLOT

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PLOT

Plots a drawing to a plotter, printer, or file.

PLOTTERMANAGER

Displays the Plotter Manager, where you can add or edit a plotter configuration.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Select a Printer or Plotter

Before plotting a drawing, you must select a printer or plotter. The device you select affects the printable area of the drawing.

After selecting a printing or plotting device, you also can easily plot a drawing using the default settings in the Plot dialog box.

To select a printer or plotter

1. Click Output tab ➤ Plot panel ➤ Plot.
2. In the Plot dialog box, under Printer/Plotter, select a plotter from the Name list.

   If you have already selected a paper size and it is not supported by the plotter you have chosen, you are warned that a paper size supported by the plotter will be used. Click OK if the warning is displayed.
3. Once you have selected a plotter, you can continue to select a paper size, or if the paper size is correct, click OK to plot the drawing.

Toolbar: Standard
Command entry: PLOT

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PLOT

Plots a drawing to a plotter, printer, or file.

PLOTTERMANAGER

Displays the Plotter Manager, where you can add or edit a plotter configuration.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Specify the Area to Plot

When plotting a drawing, you must specify the area of the drawing to plot. The Plot dialog box provides the following options under Plot Area.

- **Layout or 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 the Model tab, plots the entire drawing area 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 might be regenerated to recalculate the extents before plotting.
- **Display**. Plots the view in the current viewport in the Model tab or the current paper space view in a layout tab.
- **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. Click the Window button to use a pointing device to specify opposite corners of the area to be plotted, or enter coordinate values.

To set the drawing area as you plot

1. Click Output tab ➤ Plot panel ➤ Plot. 
2. In the Plot dialog box, under Plot Area, specify the portion of your drawing that you want to plot.

3 Change other settings as needed. Click OK to plot the drawing.

Toolbar: Standard 
Command entry: PLOT

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PLOT

Plots a drawing to a plotter, printer, or file.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Set Paper Size

In the Plot dialog box, select the paper size that you want to use.

If you plot from a layout, you may have already specified a paper size in the Page Setup dialog box. However, if you plot from the Model tab, you need to specify a paper size when you plot. In the Plot dialog box, select the paper size that you want to use. The list of paper sizes depends on the printer or plotter that you have selected in either the Plot or Page Setup dialog box. The list of available plotters includes all those that are currently configured for use with Windows and those for which you have installed non-system drivers.

You can also set the default page size used to create new layouts for most plotters by editing the PC3 file associated with that plotter. For Windows system printers, you can use this technique to specify different default page sizes for Windows and for this program.

NOTE If the PAPERUPDATE system variable is set to 0, you are prompted if the layout's existing paper size is not supported by the plotter you have selected. If the PAPERUPDATE system variable is set to 1, the paper size is automatically updated to reflect the default paper size of the selected plotter.

Use a Custom Paper Size

If you need to specify a paper size that is not listed in either the Plot dialog box or the Page Setup dialog box, you can add a custom paper size for a non-system plotter using the Plotter Configuration Editor. Typically, you cannot add a custom paper size to Windows system printers because the allowable page sizes and printable areas are determined by the manufacturer. However, you can modify the printable area for paper sizes associated with a Windows system printer.

To select a paper size for the current plot

1 Click Output tab ➤ Plot panel ➤ Plot. 

2 In the Plot dialog box, under Printer/Plotter, select a plotter in the Name box.

3 Under Paper Size, select a paper size from the list.

The paper sizes that are listed depend on the plotter you selected.

Toolbar: Standard 
Command entry: PLOT

To set the default paper size for a layout

1 Click Output tab ➤ Plot panel ➤ Page Setup. 

2 In Page Setup Manager, Page Setups area, the layout for which you want to set the paper size should be selected. If not, select the layout. Click Modify.

3 In the Page Setup dialog box, under Paper Size, select a paper size from the list. Click OK.

The paper sizes that are listed depend on the plotter specified in the page setup.

4 In Page Setup Manager, click Close.

The layout reflects the changes.

Toolbar: Layouts 
Command entry: PAGESETUP

To set the default paper size for a plotter

1 Click Output tab ➤ Plot panel ➤ Plot. 

2 In the Plot dialog box, under Printer/Plotter, select a plotter from the Name list.

3 Click Properties.

The Plotter Configuration Editor is displayed.

4 To specify a default paper size, use one of the following methods:

- For a non-system plotter, under Media, select Source and Size.
- For a system printer, in the tree view, select Custom Properties. Then under Access Custom Dialog, select Custom Properties.

5 Select the appropriate paper size.

6 Click OK to close each dialog box.

NOTE The available paper sizes depend on the printer or plotter. Some plotter manufacturers may control paper size differently in the Plotter Configuration Editor.

Toolbar: Layouts 
Command entry: PAGESETUP

To create or edit a custom paper size for a non-system printer

1 Click Output tab ➤ Plot panel ➤ Manage Plotters. 

2 In the Plotter Manager, double-click the PC3 file whose configuration you want to change.

3 In the Plotter Configuration Editor, Device and Document Settings tab, under the User Defined Paper Sizes & Calibration, select Custom Paper Sizes.

4 Set a new paper size using one of the following methods:

- To add a custom paper size, click Add and follow the steps in the Custom Paper Size wizard. You specify the size of the paper, the printable area, and a name for the new paper size.
- To edit an existing paper size, under Custom Paper Sizes, select the paper size and click Edit. The Custom Paper Size wizard opens. Change any of the paper size settings.

5 Click OK.

The new or edited paper size is available in both the Plot and Page Setup dialog boxes when that PC3 file is selected.

NOTE Creating a custom paper size for a non-system driver attaches a plot model parameter (PMP) file to the plotter configuration (PC3) file. The PMP file contains custom plotter calibration and custom paper size information. By default, PMP files are stored in the *Drv* folder.

Command entry: PLOTTERMANAGER

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PLOT

Plots a drawing to a plotter, printer, or file.

PLOTTERMANAGER

Displays the Plotter Manager, where you can add or edit a plotter configuration.

System Variables

PAPERUPDATE

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.

Utilities

No entries

Command Modifiers

No entries

Position the Drawing on the Paper

There are several ways to position a drawing on the paper. You can specify the printable area, set the position of the plot, and set the drawing orientation.

Specify the Printable Area

The printable area is displayed by a dashed border in a layout. The plotter and paper size you select determine the printable area.

WARNING If you set your plotter to use paper-saving features such as plotting inked area or nesting, your plotter will probably not use the printable area and plot offset specifications.

If your plotter reports an incorrect printable area for your paper size, you can adjust the printable area in the Modify Standard Paper Sizes area under the Modify Standard Paper Sizes (Printable Area) option on the Device and Document Settings tab in the Plotter Configuration Editor.

NOTE The Modify Standard Paper Sizes option is not a margins feature. Specify where your drawing is plotted on the page in the Plot Offset area in the Plot dialog box.

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PLOT

Plots a drawing to a plotter, printer, or file.

PLOTTERMANAGER

Displays the Plotter Manager, where you can add or edit a plotter configuration.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Set the Position of the Plot

The printable area of a drawing sheet is defined by the selected output device and is represented by the dashed line in a layout. You can change the position of plot relative to the printable area or to the edge of the paper.

The printable area of a drawing sheet is defined by the selected output device and is represented by the dashed line in a layout. When you change to another output device, the printable area may change.

The settings in the Plot Offset area of the Plot dialog box specify an offset of the plot area relative to the lower-left corner (the origin) of the printable area or 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). The Plot Offset area of the Plot dialog box displays the specified plot offset option in parentheses.

You can offset the drawing on the paper by entering positive or negative values in the X and Y offset boxes. However, this may result in the plot area being clipped. If the Plot Area is not set to Layout (Extents, Display, View, or Window), you can also select the Center the Plot option.

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PLOT

Plots a drawing to a plotter, printer, or file.

PLOTTERMANAGER

Displays the Plotter Manager, where you can add or edit a plotter configuration.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Set Drawing Orientation

The drawing orientation determines whether the position of the plotted drawing is landscape (the longer edge of the drawing is horizontal) or portrait (the longer edge of the drawing is vertical). This is based on the size of paper selected. You can also choose to plot upside down.

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PLOT

Plots a drawing to a plotter, printer, or file.

PLOTTERMANAGER

Displays the Plotter Manager, where you can add or edit a plotter configuration.

System Variables

PLOTROTMODE

Controls the orientation of plots.

Utilities

No entries

Command Modifiers

No entries

Control How Objects Are Plotted

You can control how objects are plotted by setting the plot scale, by using plot styles and plot style tables, and by setting an object's layer properties.

Set Plot Scale

When you specify a scale to output your drawing, you can choose from a list of real-world scales, enter your own scale, or select Fit to Paper to scale the drawing to fit onto the selected paper size.

Usually, you draw objects at their actual size. That is, you decide how to interpret the size of a unit (an inch, a millimeter, a meter) and draw on a 1:1 scale. For example, if your unit of measurement is millimeters, then every unit in your drawing represents a millimeter. When you plot the drawing, you either specify a precise scale or fit the image to the paper.

Most final drawings are plotted at a precise scale. The method used to set the plot scale depends on whether you plot the Model tab or a layout:

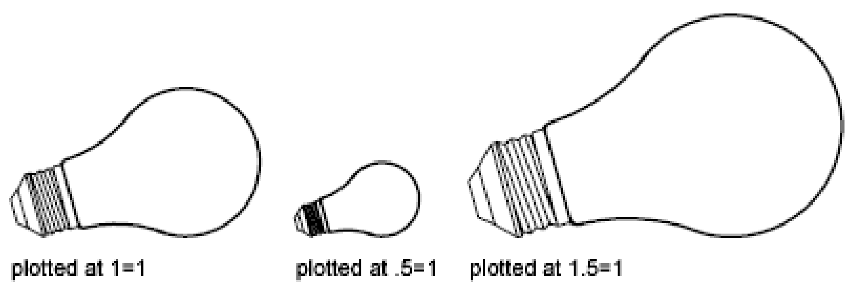
- On the Model tab, you can establish the scale in the Plot dialog box. This scale represents a ratio of plotted units to the world-size units you used to draw the model.
- In a layout, you work with two scales. The first affects the overall layout of the drawing, which usually is scaled 1:1, based on the paper size. The second is the scale of the model itself, which is displayed in layout viewports. The scale in each of these viewports represents a ratio of the paper size to the size of the model in the viewport.

NOTE You can modify the list of scales that are displayed in all view and plot scale lists with SCALELISTEDIT.

Set a Specific Scale

When you plot, the paper size you select determines the unit type, inches or millimeters. For example, if the paper size is in mm, entering **1** under mm and **10** under Units produces a plotted drawing in which each plotted millimeter represents 10 actual millimeters.

The illustrations show a light bulb plotted at three different scales.



Scale the Drawing to Fit the Page

When you review drafts, a precise scale is not always important. You can use the Fit to Paper option to plot the view at the largest possible size that fits the paper. The height or width of the drawing is fit to the corresponding height or width of the paper.

When you plot a perspective view from model space, the view is scaled to fit the paper even when you enter a scale.

When you select the Fit to Paper option, the text boxes change to reflect the ratio of plotted units to drawing units. This scale is updated whenever you change the paper size, plotter, plot origin, orientation, or size of the plotted area in the Plot dialog box.

NOTE This option is not available when the Plot Area is set to Layout.

To plot using a real-world scale

1. Click Output tab ➤ Plot panel ➤ Plot.
2. In the Plot dialog box, under Plot Scale, select a scale from the Scale box.
3. Click OK to plot the drawing.

Toolbar: Standard
Command entry: PLOT

To plot using a custom scale

1. Click Output tab ➤ Plot panel ➤ Plot.
2. In the Plot dialog box, under Plot Scale, enter a custom scale. The scale requires two values, the number of plotted units (inches or mm) per the number of drawing units. The type of unit is determined by the paper size, but you can change it in the list box.

   If you enter a custom scale, Custom is automatically selected in the Scale box, even if the scale you enter is the same as a standard scale in the list. A custom scale is the ratio between the plotted units and drawing units. For example, 1:12 and 2:24 are plotted at the same scale.
3. Click OK to plot the drawing.

Toolbar: Standard
Command entry: PLOT

To scale a drawing to fit the page

1 Click Output tab ➤ Plot panel ➤ Plot.

2 In the Plot dialog box, under Plot Scale, select the Fit to Paper option.

  The resulting scale is automatically calculated. The ratio of plotted units to drawing units in the custom scale boxes is displayed.

3 Click OK to plot the drawing.

NOTE This option is not available when the Plot Area is set to Layout.

Toolbar: Standard
Command entry: PLOT

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PLOT

Plots a drawing to a plotter, printer, or file.

SCALELISTEDIT

Controls the list of scales available for layout viewports, page layouts, and plotting.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Set Shaded Viewport Options

You can choose among several options for plotting shaded and rendered viewports. You can plot a viewport as it is displayed, in wireframe, with hidden lines removed, or as rendered.

You can specify how each viewport should be plotted, and save the plot settings with a drawing. You also can choose from a wide variety of resolutions, up to the resolution of your plotter, and save the resolution settings with a drawing.

NOTE If hardware acceleration is disabled or is enabled, but does not support all of the available hardware effects, it is possible to plot a drawing that contains shaded viewports with unsupported hardware effects by the graphics card through software emulation. To enable 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 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.

Overview of Shaded Viewport Plotting

The options for shaded viewport plotting give you a large degree of flexibility in conveying your three-dimensional designs to others. You can convey your design intent by choosing how viewports are plotted and by specifying resolution levels.

With shaded plotting options, you can choose whether to plot a set of shaded objects as displayed or in wireframe, hidden mode, a visual style, or rendered. Shaded and rendered viewports are plot-previewed, plotted, plotted to file, and published with full shading and rendering.

You can use realistic plots in your presentations by plotting viewports as they are displayed on the screen or otherwise.

Shaded viewport plotting options apply to all objects in viewports and model space. If you use the Shaded or Rendered options plot style tables included in the page setup do not affect plots. If you use the Render option, two-dimensional wireframe objects, such as lines, arcs, and text, are not plotted.

NOTE Shaded viewport plotting requires a raster-capable device. Most modern plotters and printers are raster-capable devices.

Quick Reference

Commands

3DCONFIG

Sets options that affect 3D display performance.

PLOT

Plots a drawing to a plotter, printer, or file.

SHADEMODE

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Specify Shaded Plotting Settings

If you are plotting a drawing that contains 3D solids that are shaded, you can control how the drawing is plotted.

Specifically, you can choose from the following options:

- **As Displayed.** Plots the design as it is displayed; all the shading is preserved.
- **Wireframe.** Displays lines and curves to represent object boundaries.

- **Hidden.** Suppresses the plotting of objects that are located behind other objects.
- **Visual Styles.** Plots the design as it appears in the visual style you select.
- **Rendered.** Renders objects before they are plotted, based on Render options you set before you plot or based on the render preset you select.
- **Render Presets.** Renders objects based on the render preset you select.

You can select an option for your drawing either from model space or from a layout. From model space, the options are available in the Properties palette and the Plot dialog box. From a layout, after you select a viewport, the options are available from the shortcut menu and from the Properties palette.

Either way, you can save your settings with the drawing.

NOTE If you select the Rendered option, specify Render settings before plotting. If the Rendered option is used for a highly complex set of objects, the hardcopy output might contain only the viewport border.

To modify the shade plot setting of a viewport

1. Make sure you are on a layout tab.
2. Double-click the border of the viewport you want to modify.
3. In the Properties palette, under Misc., select Shade Plot, and then select an option for plotting.

Shortcut menu: Select the viewport and right-click in the drawing area. Click a setting from the Shade Plot options.

To modify the shade plot setting in model space

1. Make sure you are on the Model tab and no objects are selected.
2. Click Output tab ➤ Plot panel ➤ Plot.
3. In the Plot dialog box, under Shaded Viewport Options, Shade Plot drop-down list, select an option.

Command entry: PLOT

Quick Reference

Commands

PLOT

Plots a drawing to a plotter, printer, or file.

3DCONFIG

Sets options that affect 3D display performance.

SHADEMODE

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Specify a Resolution Level for Shaded Plotting

You can set the resolution of shaded plots for either greater speed or higher fidelity.

After you select an appropriate plotter, you can specify the level of quality for plotted output. The quality level determines the dots per inch (dpi). The dpi that corresponds to a quality level is based on the plotter you select.

The maximum dpi available is also based on the plotter you select. You can specify a custom quality level and directly change the dpi to a setting between 100 and the maximum dpi of the plotter.

The higher the fidelity, the more computer memory is used, so the longer it takes to plot. High fidelity is not necessary for all plots, and a setting between 300 and 600 dpi is generally sufficient for most plots.

To specify a resolution level for shaded plotting

1 Click Output tab ➤ Plot panel ➤ Plot. 

2 In the Plot dialog box, under Shaded Viewport Options, Quality drop-down list, select an option.

   If you want to enter a custom dpi, select Custom and specify a dpi in the DPI box.

3 Click OK.

Command entry: PLOT

Quick Reference

Commands

PLOT

Plots a drawing to a plotter, printer, or file.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Set Options for Plotted Objects

In the Plot and the Page Setup dialog boxes, you can choose from options that affect how objects are plotted.

- **Shaded Viewport Plotting.** Specifies shaded plotting options: As Displayed, Wireframe, or Hidden. The effect of this setting is reflected in the plot preview, but not in the layout.
- **Plot Object Lineweights.** Specifies that lineweights assigned to objects and layers are plotted.
- **Plot with Plot Styles.** Specifies that the drawing is plotted using plot styles. Selecting this option automatically plots lineweights. If you do not select this option, objects are plotted with their assigned properties and not with the plot style overrides.

NOTE Plot styles are not available for objects with the Jitter edge modifier applied (VISUALSTYLES).

- **Plot Paper Space Last.** Specifies that objects in model space are plotted before those in paper space.
- **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 and 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, where you can specify the information 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 Plot Stamp On in the Plot dialog box, and then click the Plot Stamp Settings button.
- **Save Changes to Layout.** Saves changes you make in the Plot dialog box to the layout if you click OK.

To set shaded plotting options from the Model tab

1 Click Output tab ➤ Plot panel ➤ Plot.

2 In the Plot dialog box, Shaded Viewport Options area, Shade Plot drop-down list, select an option.

Toolbar: Standard
Command entry: PLOT

To set shaded plotting options from a layout tab

1 Select a layout viewport.

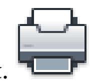
2 Double-click the viewport border to display the Properties palette.

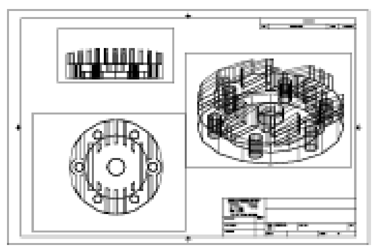
3 On the Properties palette, click Shade Plot.

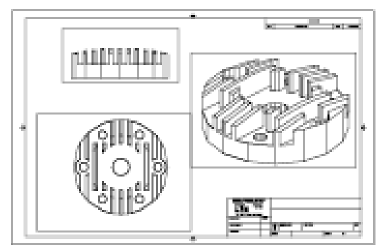
4 In the Shade Plot drop-down list, select an option.

Toolbar: Standard 
Command entry: PROPERTIES
Shortcut menu: Select the viewport, right-click in the drawing area, and then click Shade Plot.

To remove hidden lines when plotting from the Model tab

1. Click Output tab ➤ Plot panel ➤ Plot. 
2. In the Plot dialog box, under Shaded Viewport Options area, Shade Plot drop-down list, select Hidden.
3. Click OK to plot the drawing.

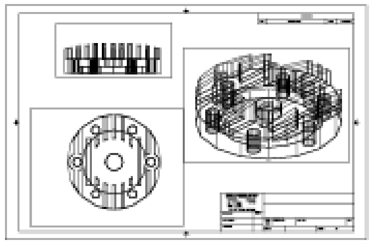

Hidden lines not removed


Hidden lines removed

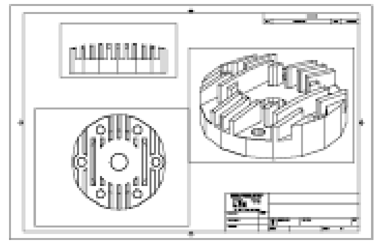
Toolbar: Standard 
Command entry: PLOT

To remove hidden lines when plotting from a layout tab

1. Select a layout viewport.
2. Double-click the viewport border to display the Properties palette.
3. On the Properties palette, click Shade Plot.
4. In the Shade Plot drop-down list, select Hidden.
5. Click OK to plot the drawing.



Hidden lines not removed



Hidden lines removed

Toolbar: Standard 
Command entry: PROPERTIES
Shortcut menu: Select the viewport and right-click in the drawing area. Click Shade Plot.

To plot lineweights

1. Click Output tab ➤ Plot panel ➤ Plot. 
2. In the Plot dialog box, under Plot Options, select Plot Object Lineweights.

   You can change this option only if the Plot with Plot Styles option is cleared.
3. Click OK to plot the drawing.

Toolbar: Standard 
Command entry: PLOT

To turn off plot styles

1. Click Output tab ➤ Plot panel ➤ Plot. 
2. In the Plot dialog box, under Plot Options, clear the Plot with Plot Styles option.
3. Click OK to plot the drawing.

Toolbar: Standard
Command entry: PLOT

To change the order in which objects are plotted

1 Click Output tab ➤ Plot panel ➤ Plot.
2 In the Plot dialog box, under Plot Options, select Plot Paperspace Last. Clear the Plot Paperspace Last option to plot paper space first.
3 Click OK to plot the drawing.

Toolbar: Standard
Command entry: PLOT

To hide paper space objects when plotting from a layout tab

1 Click Output tab ➤ Plot panel ➤ Plot.
2 In the Plot dialog box, under Plot Options, select Hide Paperspace Objects.
3 Click OK to plot the drawing.

Toolbar: Standard
Command entry: PLOT

Quick Reference

Commands

LWEIGHT

Sets the current lineweight, lineweight display options, and lineweight units.

OPTIONS

Customizes the program settings.

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PLOT

Plots a drawing to a plotter, printer, or file.

PLOTSTAMP

Places a plot stamp on a specified corner of each drawing and logs it to a file.

PROPERTIES

Controls properties of existing objects.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Use Plot Styles to Control Plotted Objects

You can control many aspects of how an object is plotted by using plot styles.

Overview of Plot Styles

A plot style controls an object's plotted properties.

A plot style is an object property, similar to linetype and color. A plot style can be assigned to an object or assigned to a layer. A plot style controls an object's plotted properties, including

- Color
- Dither
- Grayscale
- Pen number

- Virtual pen
- Screening
- Linetype
- Lineweight
- Line end style
- Line join style
- Fill style

Using plot styles gives you great flexibility because you can set them to override other object properties or turn off the override as needed.

Groups of plot styles are saved in either of two types of plot style tables: color-dependent (CTB) or named (STB). Color-dependent plot style tables set style based on the color of the object. Named plot styles can be assigned to an object independent of color.

NOTE Plot styles are not available for objects with the Jitter edge modifier applied (VISUALSTYLES).

Quick Reference

Commands

CONVERTCTB

Converts a color-dependent plot style table (CTB) to a named plot style table (STB).

CONVERTPSTYLES

Converts the current drawing to either named or color-dependent plot styles.

OPTIONS

Customizes the program settings.

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

System Variables

CPLOTSTYLE

Controls the current plot style for new objects.

DEFLPLSTYLE

Specifies the default plot style for all layers in a drawing when 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.

DEFPLSTYLE

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.

PSTYLEMODE

Indicates whether the current drawing is in a Color-Dependent or Named Plot Style mode.

PSTYLEPOLICY

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.

Utilities

No entries

Command Modifiers

No entries

Choose a Type of Plot Style Table

A plot style table is a collection of plot styles assigned to a layout or the Model tab. There are two types of plot style tables: color-dependent plot style tables and named plot style tables.

Color-dependent plot style tables (CTB) use an object's color to determine characteristics such as lineweight. Every red object in a drawing is plotted the same way. While you can edit plot styles in a color-dependent plot style table, you cannot add or delete plot styles. There are 256 plot styles in a color-dependent plot style table, one for each color.

Named plot style tables (STB) contain user-defined plot styles. When you use a named plot style table, objects that have the same color may be plotted differently, based on the plot style assigned to the object. A named plot style table can contain as many or as few plot styles as required. Named plot styles can be assigned to objects or layers, just like any other property.

To set a plot style table type for new drawings

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Plot and Publish tab, click the Plot Style Table Settings button.
3. In the Plot Style Table Settings dialog box, select Use Color-dependent Plot Styles or Use Named Plot Styles.
4. (Optional) In the Default Style Table box, select a default plot style table.
5. (Optional) If Use Named Plot Styles is selected, select plot styles to assign to Layer 0 and to new objects.
6. Click OK.

NOTE Setting plot style table types for new drawings does not affect existing drawings.

Command entry: OPTIONS

Quick Reference

Commands

OPTIONS

Customizes the program settings.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Switch the Type of Plot Style Table

You can change which type of plot style table, color-dependent or named, is used in a drawing.

You can use CONVERTPSTYLES to change which type of plot style table, color-dependent or named, is used in a drawing.

When you convert a drawing from using color-dependent plot style tables to using named plot style tables, any color-dependent plot style tables attached to layouts in the drawing are removed, and named plot styles are attached in their place. If you want to use the styles defined in the color-dependent plot style tables after you convert to using named plot styles tables, you should first convert any color-dependent plot style tables to named plot style tables.

When you convert a drawing from using named plot style tables to using color-dependent plot style tables, the plot style names assigned to the objects in the drawing are lost.

In addition to changing which type of plot style table a drawing uses, you can use CONVERTCTB to convert color-dependent plot style tables to named plot style tables. You cannot, however, convert a named plot style table to a color-dependent plot style table.

To convert a drawing to use named plot styles

1. At the command prompt, enter **convertpstyles**.
2. Click OK when the alert box appears.
3. In the Select File dialog box, select a named plot style table to use for the Model tab and any layouts that use plot style tables of the same name.
4. Click Open.

   A message is displayed to confirm that the drawing was converted.

NOTE If the Display Plot Styles option was already selected in the dialog box, you must enter **regen** at the command prompt to display the plot style settings.

To convert a drawing to use color-dependent plot styles

1. At the command prompt, enter **convertpstyles**.
2. Click OK.

   A message is displayed to confirm that the drawing was converted.

To convert a color-dependent plot style table into a named plot style table

1. At the command prompt, enter **convertctb**.
2. In the Select File dialog box, select the name of the plot style table to convert, and then click Open. By default, plot style tables are saved in the *Plot Styles* folder.
3. Enter the new plot style table name. Click Save.
4. Click OK when the alert box appears.

NOTE Be sure to use the Plot Style Table Editor to change the names of the plot styles in their new plot style table to be more meaningful *before* using the plot style table with any drawings.

Command entry: CONVERTCTB

Quick Reference

Commands

CONVERTCTB

Converts a color-dependent plot style table (CTB) to a named plot style table (STB).

CONVERTPSTYLES

Converts the current drawing to either named or color-dependent plot styles.

OPTIONS

Customizes the program settings.

OPTIONS

Customizes the program settings.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Assign Plot Style Tables to Layouts

By assigning different plot style tables to each layout in your drawing, you can control how objects in the layout are plotted. The plot style table affects both model space and paper space objects. To plot the drawing without applying plot style properties, select None from the list of plot style tables.

If you use named plot style tables, each object in the drawing either is assigned a plot style directly or inherits a plot style from its layer.

To display the effects of a plot style table in a layout, select Display Plot Styles under Plot Style Table in the Page Setup dialog box.

NOTE If you insert an xref into your current drawing, all defined plot style tables are also inserted. You can modify the appearance of your objects by editing the attached plot style tables with the Plot Style Table Editor.

To assign a plot style table to a layout

1 Click the Model tab or the layout tab to which you want to assign the plot style table.

2 Click Output tab ➤ Plot panel ➤ Page Setup. 

3 In Page Setup Manager, click Modify.

4 Under Plot Style Table (Pen Assignments), select a plot style table from the list.

5 In the Question dialog box, click Yes or No to indicate whether the selection should be applied to the current tab only or to all layouts.

   This option is available only for the Model tab.

6 To preview the effects of the plot style table in the layout, select Display Plot Styles.

   This option is available only for layouts.

7 Click OK.

8 In Page Setup Manager, click Close.

NOTE If the Display Plot Styles option was already selected in the dialog box, you must enter **regen** at the command prompt to display the plot style settings.

Toolbar: Layouts 
Command entry: PAGESETUP

To preview the effects of a plot style table in a layout

1 Click the layout tab in which you want to preview the effects of the plot style table.

2 Click Output tab ➤ Plot panel ➤ Page Setup. 

3 In Page Setup Manager, click Modify.

4 In the Page Setup dialog box, under Plot Style Table (Pen Assignments), select the Display Plot Styles option.

5 Click OK.

6 In Page Setup Manager, click Close.

   The effects of the plot style table are previewed in the layout.

Toolbar: Layouts 
Command entry: PAGESETUP

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Manage Plot Style Tables

You can use the Plot Style Manager to add, delete, rename, copy, and edit plot style tables.

Color-dependent (CTB) and named (STB) plot style tables are stored in the *Plot Styles* folder by default. This folder is also known as the Plot Style Manager.

You can use the Plot Style Manager to add, delete, rename, copy, and edit plot style tables. The Plot Style Manager lists all of the available plot style tables.

To create a plot style table

1. Click Tools menu ➤ Wizards ➤ Add Plot Style Table.
2. Read the first page. Click Next.
3. On the Begin page, you can choose to use an configuration file (CFG) or plotter configuration file (PCP or PC2) to import pen settings, base the new plot style table on an existing plot style table, or start from scratch. If you use an existing plot style table, the new plot style table uses the same type of table as the original. Click Next.
4. On the Pick Plot Style Table page, select Color-Dependent Plot Style Table or Named Plot Style Table.

5 If you are importing pen settings from a PCP, PC2, or CFG file, or if you are basing the new plot style table on an existing plot style table, specify the file on the Browse File Name page. If you use a CFG file, you may need to select which plotter's configuration to import. Click Next.

6 On the File Name page, enter a name for the new plot style table. Click Next.

7 On the Finish page, you can edit the new plot style table by choosing Plot Style Table Editor. You can assign the new plot style table so that it can be used in all drawings.

8 Click Finish.

   The new plot style table is available in both the Plot and the Page Setup dialog boxes for all drawings using color-dependent plot style tables.

To rename a plot style table

1 Click Output tab ➤ Plot panel ➤ Plot Style Manager. 

2 Right-click the plot style whose name you want to change. Click Rename.

3 Enter the new file name. Be sure to add the same file extension (*.ctb* or *.stb*). When finished, press ENTER.

   The renamed plot style table is available in the Plot and Page Setup dialog boxes for all drawings using that type of plot style table.

Command entry: STYLESMANAGER

To change a plot style table description

1 Click Output tab ➤ Plot panel ➤ Plot Style Manager. 

2 Double-click the plot style table file whose description you want to change.

3 In the Plot Style Table Editor, General tab, enter the new description for the plot style table.

4 Click Save & Close.

Command entry: STYLESMANAGER

To edit plot styles in a plot style table

1 Click Output tab ➤ Plot panel ➤ Plot Style Manager.

2 Double-click the plot style table you want to change.

3 In the Plot Style Table Editor, Form View tab, under Plot Styles, select a plot style and edit the settings.

4 Click Save & Close.

Command entry: STYLESMANAGER

Quick Reference

Commands

STYLESMANAGER

Displays the Plot Style Manager, where you can revise plot style tables.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Use Color-Dependent Plot Style Tables

By using color-dependent plot styles to control how objects are plotted, you ensure that all objects that share the same color are plotted the same way.

When a drawing uses color-dependent plot style tables, you cannot assign a plot style to individual objects or layers. Instead, to assign plot style properties to an object, you change the color of the object or layer.

You can assign color-dependent plot style tables to layouts. You can use several predefined color-dependent plot style tables, edit existing plot style tables, or create your own.

Color-dependent plot style tables are stored in the *Plot Styles* folder and have a *.ctb* extension.

Use Predefined Color-Dependent Plot Style Tables

Several color-dependent plot style tables are installed in the *Plot Styles* folder, also known as the Plot Style Manager.

Table	Description
acad.ctb	Default plot style table
fillPatterns.ctb	Sets first 9 colors to use first 9 fill patterns, all others to use object's fill
grayscale.ctb	Converts all colors to grayscale when plotted
monochrome.ctb	Plots all colors as black
None	Applies no plot style table
screening 100%.ctb	Uses 100% ink for all colors
screening 75%.ctb	Uses 75% ink for all colors
screening 50%.ctb	Uses 50% ink for all colors
screening 25%.ctb	Uses 25% ink for all colors

NOTE You can assign a color-dependent plot style table to a layout only if the drawing has been set to use color-dependent plot style tables.

See also:

- Assign Plot Style Tables to Layouts on page 1820

Quick Reference

Commands

STYLESMANAGER

Displays the Plot Style Manager, where you can revise plot style tables.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Use Named Plot Style Tables

You can only create, delete, and apply plot styles in a named plot style table. You can define as many or as few plot styles as you need in a drawing.

Use Named Plot Styles

Named plot styles are assigned to objects and layers in the same way that linetype and color are assigned to objects.

An object whose plot style is set to BYLAYER inherits the plot style assigned to its layer.

Use the Properties palette to change an object's plot style and the Layer Properties Manager to change the plot style for a layer.

Because different plot style tables can be assigned to each layout and a named plot style table can contain any number of plot styles, an object or layer may have a plot style assigned to it that is not in every plot style table. In this case, the plot style as missing in the Select Plot Style dialog box; the object's default plotting properties are used. For example, named plot style table Style1 contains plot styles A and B. Named plot style table Style2 contains plot styles B and C. In a layout that uses Style1, any objects that use plot style C are listed as having a missing plot style. Objects that are assigned plot style C in this layout are plotted using their default settings.

To change an object's plot style

You can change an object's plot style only if the drawing uses named plot style tables. If the drawing uses color-dependent plot style tables, change the object's color to alter its plotted appearance.

1 Select one or more objects whose plot style you want to change.

2 Click Veiw tab ➤ Palettes panel ➤ Properties.

TIP You can also right-click in the drawing area and then, click Properties.

3 In the Properties palette, select a plot style from the list of available plot styles. Click the column next to Plot Style.

   The plot styles listed are those already in use by objects and in the plot style table attached to the current layout.

4 To select a plot style from a different plot style table, select Other. In the Select Plot Style dialog box, you can attach a different plot style table to the current layout and select a plot style from that plot style table.

5 To edit the current plot style table, click Editor.

6 Click OK when finished.

   Any changes made in the Properties palette are immediate. If you change the plot style table attached to the current layout, both model space and paper space objects are affected.

Toolbar: Standard
Command entry: PROPERTIES

To change a layer's plot style

1 Click Home tab ➤ Layers panel ➤ Layer Properties Manager.

2 In the Layer Properties Manager, select the layer whose plot style you want to change.

3 Click the current plot style in the Plot Style column. Select the plot style you want to use.

4 To select a plot style from a different plot style table, select an active plot style table from the Active Plot Style Tables list.

The list of plot styles changes to those in the selected plot style table.

5 To edit the selected plot style table, click Editor. Change settings as necessary and click Save & Close.

6 Click OK.

You can change a layer's plot style only if the drawing uses named plot style tables. If the drawing uses color-dependent plot style tables, change the layer's color to alter the plotted appearance of objects on the layer.

NOTE An object's plot style property can be set to BYLAYER to inherit the plot style of its layer.

Toolbar: Layers 
Command entry: LAYER

To set the current plot style

1 Click Home tab ➤ Properties panel ➤ Plot Style. 

2 In the Current Plot Style dialog box, select a plot style from the list.

The plot styles displayed are those available in the current plot style table.

3 To select a plot style from a different plot style table, under Active Plot Style Table, select a plot style table.

The list of plot styles changes to those in the selected plot style table.

4 To edit the selected plot style table, click Editor. Change settings as necessary. Click Save & Close.

5 Click OK.

The plot style that you made current is used for any new objects created in the drawing.

You can change the current plot style only if the drawing uses named plot style tables. If the drawing uses color-dependent plot style tables, change the color to alter the plotted appearance of objects and layers.

NOTE The current plot style property can be set to BYLAYER to inherit the plot style of the current layer.

Command entry: PLOTSTYLE

Quick Reference

Commands

LAYER

Manages layers and layer properties.

PLOTSTYLE

Controls the named plot styles that are attached to the current layout and can be assigned to objects.

PROPERTIES

Controls properties of existing objects.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Manage Named Plot Styles

You can add, delete, rename, and copy plot styles in a named plot style table using the Plot Style Table Editor.

The Plot Style Table Editor is also used to change plot style settings for both named and color-dependent plot style tables.

NOTE You cannot delete or edit the NORMAL plot style. Also, you cannot add, delete, copy, or rename plot styles in a named plot style table if a color mapping table has been attached to the plot style table. A color mapping table associates every plot style with an ACI color.

See also:

- Change Plot Style Settings on page 1835

To create a named plot style

1 Click Home tab ➤ Properties panel ➤ Plot Style.

2 Double-click the STB file to which you want to add a plot style.

3 (Optional) If you want to position the plot style in the list, on the Form View tab, select the plot style that should precede the new plot style.

4 On the Form View or Table View tab, click Add Style.

5 In the Add Plot Style dialog box, enter the name of the plot style.

6 Click Save & Close.

NOTE You cannot edit the NORMAL plot style.

Command entry: STYLESMANAGER

To copy a named plot style

1 Click Home tab ➤ Properties panel ➤ Plot Style.

2 Double-click the STB file you want to edit.

3 In the Plot Style Table Editor, Form View tab, right-click the plot style that you want to copy. Click Copy.

4 Right-click a plot style. Click Paste. If you want to position the new plot style in the list, right-click the plot style that should precede the new plot style.

5 In the Add Plot Style dialog box, enter the name of the plot style.

6 Click Save & Close.

NOTE You cannot copy plot styles if the plot style table uses a color mapping table. You cannot copy plot styles in a color-dependent plot style table.

Command entry: STYLESMANAGER

To change a plot style's description

1 Click Home tab ➤ Properties panel ➤ Plot Style.

2 Double-click the plot style table that contains the plot style whose description you want to change.

3 In the Plot Style Table Editor, Form View tab, select the plot style whose description you want to change.

4 Under Description, change the plot style's description.

 You can select additional plot styles and modify their descriptions or settings.

5 Click Save & Close.

NOTE You cannot delete or edit the NORMAL plot style.

Command entry: STYLESMANAGER

To rename a named plot style

1 Click Home tab ➤ Properties panel ➤ Plot Style.

2 Double-click the STB file that contains the plot style you want to rename.

3 In the Plot Style Table Editor, Form View tab, right-click the plot style whose name you want to change. Click Rename.

4 Enter the new name for the plot style.

5 Click Save & Close.

NOTE You cannot rename the NORMAL style. You cannot rename plot styles in a color-dependent plot style table.

Command entry: STYLESMANAGER

To delete a named plot style

1. Click Home tab ➤ Properties panel ➤ Plot Style.
2. Double-click the STB file you want to edit.
3. In the Plot Style Table Editor, Form View tab, select the plot style that you want to delete from the list of plot styles.
4. Click Delete Style.
5. Click Save & Close.

NOTE You cannot delete or edit the NORMAL plot style. You cannot delete plot styles if the plot style table uses a color mapping table. You cannot delete plot styles from a color-dependent plot style table.

Command entry: STYLESMANAGER

Quick Reference

Commands

PLOTSTYLE

Controls the named plot styles that are attached to the current layout and can be assigned to objects.

STYLESMANAGER

Displays the Plot Style Manager, where you can revise plot style tables.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Use Predefined Named Plot Style Tables

One additional named plot style table is installed for you to use beyond the default plot style table. All named plot style tables have an *.stb* extension.

- *acad.stb*: Default plot style table
- *Monochrome.stb*: All colors plot as black
- None: No plot style table applied

NOTE Named plot style tables are available only if the drawing has been set to use named plot style tables.

See also:

- Assign Plot Style Tables to Layouts on page 1820

Quick Reference

Commands

PLOTSTYLE

Controls the named plot styles that are attached to the current layout and can be assigned to objects.

STYLESMANAGER

Displays the Plot Style Manager, where you can revise plot style tables.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Delete Color Mapping Tables

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.

Named plot style tables that you create using CFG, PCP, or PC2 files have color mapping tables that are created from your previous 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. This enables you to simulate the way drawings were plotted in previous versions.

While the color mapping table exists, you cannot add, delete, or rename plot styles in that plot style table.

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.

WARNING If you delete a color mapping table, plot styles cannot automatically be assigned to objects when older drawings are opened for the first time.

To delete a color mapping table

1 Click Output tab ➤ Plot panel ➤ Plot Style Manager. 

2 Double-click the plot style table that you want to modify.

3 In the Plot Style Table Editor, General tab, click Delete AutoCAD Release 14 Color Mapping Table.

4 Read the warning. Click Yes to delete the color mapping table, or click No to keep it.

5 Click Save & Close.

Command entry: STYLESMANAGER

Quick Reference

Commands

STYLESMANAGER

Displays the Plot Style Manager, where you can revise plot style tables.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Change Plot Style Settings

You can modify plot styles using the Plot Style Table Editor. Changes you make to a plot style affect the objects to which that plot style is assigned.

Overview of Plot Style Settings

You can open the Plot Style Table Editor by double-clicking a CTB or STB file in the Plotter Manager. The Plot Style Table Editor displays the plot styles contained in the specified plot style table.

The General tab lists general information about the table. The Table View and Form View tabs provide two ways to modify 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 tab might be more convenient.

In a named plot style table, the NORMAL plot style represents an object's default properties (no plot style applied). You cannot modify or delete the NORMAL style.

To edit plot style settings

1. Click Output tab ➤ Plot panel ➤ Plot Style Manager.
2. Double-click the plot style table you want to modify.
3. In the Plot Style Table Editor, Form View tab, under Plot Styles, select the plot style you want to modify.
4. Under Properties, click the arrow next to the property you want to change. Select an option from the list.
5. Edit other properties or plot styles as needed.
6. Click Save & Close.

Command entry: STYLESMANAGER

Quick Reference

Commands

STYLESMANAGER

Displays the Plot Style Manager, where you can revise plot style tables.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Set Color, Screening, Grayscale, and Dither in Plot Style Tables

You can use a plot style to assign color, screening, grayscale, and dither properties.

Assign Plot Style Colors

The default setting for plot style color is Use Object Color. With this setting, the object retains its layer or individually set color. If you assign a plot style color, the color overrides the object's color at plot time. You can specify one of 255 ACI colors, a true color, or a color book color. The plotter must be configured for True Color if you want to plot True Color plot styles.

NOTE If you use a plot style table saved in AutoCAD 2000 or later, the True Color values change to the nearest match in the current version's palette.

Use Screening

You can select 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. Screening is effective only if your plotter is configured to plot colors or grayscale. Also, dithering must be enabled.

Use Dithering

A plotter uses dithering to approximate colors with dot patterns, giving the impression of plotting with more colors than the ink colors available in the plotter. If the plotter does not support dithering, the dithering setting is ignored.

The most common reason for turning off dithering is to avoid false line typing from dithering of thin vectors and to make dim colors more visible. When you turn off dithering, colors are mapped to the nearest color, which limits the range of colors used for plotting. Dithering is available whether you use the object's color or assign a plot style color.

NOTE Dithering disables merge control.

Convert to Grayscale

When you select Convert to Grayscale, the object's colors are converted to grayscale if the plotter supports grayscale. Light colors, such as yellow, are plotted with light gray values. Dark colors are plotted with dark gray values. If you clear Convert to Grayscale, the RGB values are used for the object's colors. Conversion to grayscale is available whether you use the object's color or assign a plot style color.

To assign a plot style color

1. Click Output tab ➤ Plot panel ➤ Plot Style Manager.
2. Right-click a CTB or STB file. Click Open.
3. In the Plot Style Table Editor, Table View tab, click the Color field for the plot style you want to change.
4. On the Color drop-down list, click the color you want to use or click Select Color to display the Select Color dialog box and do one of the following:
   - On the Index tab, click a color or enter the ACI color number (1-255) or name in the Color box. Click OK.
   - On the True Color tab, in the Color Model box, specify a color. (Enter a color value in the Color box or specify values in the Hue, Saturation, and Luminance boxes.) Click OK.
   - On the Color Books tab, in the Color Book box, select a color (use the up and down arrow and click on a color chip.) Click OK.

Command entry: STYLESMANAGER

To use screening

1. Click Output tab ➤ Plot panel ➤ Plot Style Manager.
2. Double-click the plot style table that you want to modify.
3. In the Plot Style Table Editor, Form View tab, select the plot style you want to change.
4. In the Screening box, enter an intensity value between 1 and 100.
5. When finished, click Save & Close.

   You can edit properties for multiple plot styles while in the Plot Style Table Editor.

   NOTE A common practice when using screening is to set the plotted color to black for each style so that the screening will be a percentage of black.

Command entry: STYLESMANAGER

To enable or disable dithering

1 Click Output tab ➤ Plot panel ➤ Plot Style Manager.

2 Double-click the plot style table that you want to modify.

3 In the Plot Style Table Editor, Form View tab, select the plot style you want to change and select Dither.

4 Select On or Off.

5 When finished, click Save & Close.

You can edit properties for multiple plot styles while in the Plot Style Table Editor.

Command entry: STYLESMANAGER

To enable or disable conversion to grayscale

1 Click Output tab ➤ Plot panel ➤ Plot Style Manager.

2 Double-click the plot style table that you want to modify.

3 In the Plot Style Table Editor, Form View tab, select the plot style you want to change and select Grayscale.

4 Select On or Off.

5 When finished, click Save & Close.

You can edit properties for multiple plot styles while in the Plot Style Table Editor.

Command entry: STYLESMANAGER

Quick Reference

Commands

STYLESMANAGER

Displays the Plot Style Manager, where you can revise plot style tables.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Assign Pen Numbers and Virtual Pens in Plot Style Tables

The pen assignments used by the plotter differ depending on whether you use a pen plotter or another kind of printer, such as a laser printer.

Assign Pens to Plot Styles

The Use Assigned Pen Number setting in the Plot Style Table Editor specifies which physical pen to use for each plot style. The physical pens in the plotter are described in the plotter's configuration (PC3) file. If you use a pen plotter, you must provide information about the color, speed, and width of each pen in the Physical Pen Configuration section of the Plotter Configuration Editor.

For example, when you specify pen information in the Plotter Configuration Editor, you might specify that pen #1 is black and 0.010 inches and pen #2 is red and 0.020 inches. In the Plot Style Table Editor you can assign pen #1 to the plot style called WATER PIPES and pen #2 to the plot style called SEWER PIPES.

You can assign a pen to a plot style by selecting from a range of 32 pen numbers in the Use Assigned Pen Number field. The default value is 1. If plot style color is set to Use Object Color, or if you are editing a plot style in a color-dependent plot style table, you cannot change the assigned pen number.

If you specify 0, the field is updated to read Automatic. The information you provided under Physical Pen Characteristics in the Plotter Configuration Editor is used to select the pen closest in color to the color of the object you are plotting.

Assign Pen Settings to Plotters Without Pens

Many plotters that do not use pens can simulate the performance of a pen plotter by using virtual pens. For many devices, you can control the virtual pens in the device with software or, by configuring them from the plotter's control panel, with hardware.

If you allow software to control the pens, the Plot Style Table values for the Lineweight, Linetype, Screening, Line End Style, Line Join Style, and Fill Style settings are effective and override the settings on the plotter's control panel.

If you turn off software control of the pen attributes (typically done on the plotter), then the software can select virtual pens but can't control lineweight, linetype, end style, join style, fill style, or color. In the program, you select hardware (virtual pen) control over software (normal) control by selecting 255 Virtual Pens in the Color Depth area of the Vector Graphics option on the Device and Document Settings tab in the Plotter Configuration Editor. Selecting any other color depth specifies software control.

In the Plot Style Table Editor under Virtual Pen #, you can specify a virtual pen number between 1 and 255. Enter **0** or **Automatic** to specify that the virtual pen assignment should be made from the ACI.

When you create a plot style table, it is important to remember that it can be used with many different plotters and that the plotter and mode determine what parts of the plot style table are enabled.

- When using a pen plotter with user assigned pens, the virtual pen number and any color assignments are ignored.
- When using a pen plotter with automatically assigned pens, pens are selected based on entity color and entity lineweight. Virtual pen numbers are ignored.
- When using a raster plotter in raster mode, the physical pen number and the virtual pen number are ignored.
- When using a raster plotter in virtual pen mode, everything except the virtual pen number is ignored.

NOTE If you use another application to process your plot files after creating them, and you modify the pen attributes, plotting without using virtual pens results in pen numbers in the plot file having no simple relationship to object colors in the program. This makes it difficult to apply additional pen attributes.

To specify a virtual pen number

1 Click Output tab ➤ Plot panel ➤ Plot Style Manager.

2 Double-click the plot style table that you want to modify.

3 In the Plot Style Table Editor, Form View tab, select the plot style you want to change and click Virtual Pen.

4 Under Virtual Pen #, enter a number between 1 and 255 or enter **0** or **Automatic** to have the program assign the ACI color of the object you are plotting to the virtual pen.

5 When finished, click Save & Close.

You can edit properties for multiple plot styles while in the Plot Style Table Editor.

Command entry: STYLESMANAGER

Quick Reference

Commands

STYLESMANAGER

Displays the Plot Style Manager, where you can revise plot style tables.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Control Plotted Lineweight and Linetype

Both lineweight and linetype can be set as an object property or controlled when plotted by using a plot style. Lineweight or linetype settings in the plot style override the object's lineweight or linetype at plot time.

Assign and Display Lineweights

When you select the Lineweight field in the Plot Style Table Editor, a sample of the lineweight as well as its numeric value are displayed. The default setting for plot style lineweight is Use Object Lineweight. You can modify an existing lineweight if the one you need is not available.

To view plot style lineweights in a layout, select Display Plot Styles under Plot Style Table in the Page Setup dialog box.

Assign Linetypes

When you select the Linetype field in the Plot Style Table Editor, a list with a sample and a description of each linetype are displayed. The default setting for plot style linetype is Use Object Linetype.

Whether you choose to assign a linetype as a property of the object or as a plot style, you can set the Adaptive Adjustment option. This option 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.

You can apply a global scale factor to non-ISO linetypes and fill patterns in plot styles.

See also:

- Work with Linetypes on page 601
- Control Lineweights on page 614

To set the plotted lineweight

1. Click Output tab ➤ Plot panel ➤ Plot Style Manager. 
2. Double-click the plot style table that you want to modify.
3. In the Plot Style Table Editor, Form View tab, select the plot style you want to change.
4. Click the Lineweight arrow and select a lineweight from the list. To specify that the object's lineweight should be used select Use Object Lineweight.
5. Click Save & Close.

You can edit properties for multiple plot styles while in the Plot Style Table Editor.

Command entry: STYLESMANAGER

To set the plotted linetype

1 Click Output tab ➤ Plot panel ➤ Plot Style Manager. 

2 Double-click the plot style table that you want to modify.

3 In the Plot Style Table Editor, Form View tab, select the plot style you want to change.

4 Click the Linetype arrow and select a linetype from the list. To specify that the object's linetype should be used, select Use Object Linetype.

5 To adjust the linetype scale to show the complete pattern, on the Form View tab, select On in the Adaptive box.

6 Click Save & Close.

You can edit properties for multiple plot styles while in the Plot Style Table Editor.

Command entry: STYLESMANAGER

To apply scaling to non-ISO linetypes and fill patterns

1 Click Output tab ➤ Plot panel ➤ Plot Style Manager. 

2 Double-click the plot style table that you want to modify.

3 In the Plot Style Table Editor, General tab, select Apply Global Scale Factor to Non-ISO Linetypes.

This option scales linetypes and fill patterns according to a value that you specify.

4 In the Scale Factor box, enter a scale factor to apply.

5 Click Save & Close.

Command entry: STYLESMANAGER

Quick Reference

Commands

STYLESMANAGER

Displays the Plot Style Manager, where you can revise plot style tables.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Assign Plotted Line End and Join Styles

You can set the line end and join styles for objects that have lineweight assigned, either as an object property or as a plot style override.

Assign Line End Style

The program includes the following line end style options:

- Butt
- Square
- Round
- Diamond

The default setting for Line End Style is Use Object End Style, which is rounded. Assign a line end style in a plot style to override the object's default line end style at plot time.

NOTE SHX text plots best with the Round End and Round Join styles.

Assign Line Join Style

The program includes the following line join style options:

- Miter
- Bevel
- Round
- Diamond

The default setting for Line Join Style is Use Object Join Style, which is rounded. Assign a line join style in a plot style to override the object's default line join style at plot time.

To assign a line end or line join style

1. Click Output tab ➤ Plot panel ➤ Plot Style Manager. 
2. Right-click a CTB or STB file. Click Open.
3. In the Plot Style Table Editor, Table View tab, click the Line End Style or Line Join Style field for the plot style you want to change.
4. Select an option from the drop-down list.

Command entry: STYLESMANAGER

Quick Reference

Commands

STYLESMANAGER

Displays the Plot Style Manager, where you can revise plot style tables.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Assign Plotted Fill Styles

You can assign a variety of fill style options when plotting wide polylines, donuts, objects hatched with a solid fill, and solids.

The program includes the following fill style options when plotting wide polylines, donuts, objects hatched with a solid fill, and solids:

- Solid
- Checkerboard
- Crosshatch
- Diamonds
- Horizontal Bars
- Slant Left
- Slant Right
- Square
- Dots
- Vertical Bar

The default setting for Fill Style is Use Object Fill Style. Assign a fill style in a plot style to override the object's fill style at plot time.

You can apply a global scale factor to non-ISO linetypes and fill patterns in plot styles.

See also:

- Choose Hatch Patterns and Solid Fills on page 1449

To assign a fill style

1 Click Output tab ➤ Plot panel ➤ Plot Style Manager.
2 Right-click a CTB or STB file. Click Open.
3 In the Plot Style Table Editor, Table View tab, click the Fill Style field for the plot style you want to change.
4 Select a fill style from the drop-down list.

Command entry: STYLESMANAGER

Quick Reference

Commands

STYLESMANAGER

Displays the Plot Style Manager, where you can revise plot style tables.

System Variables

No entries

Utilities

No entries

Command Modifiers

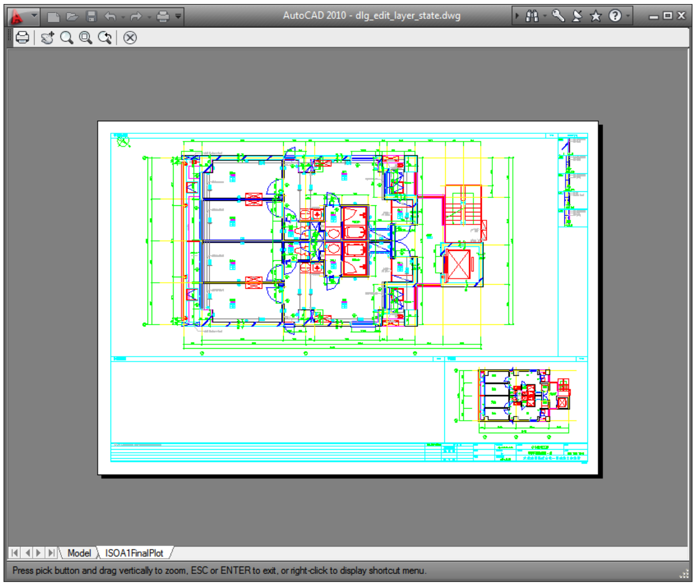
No entries

Preview a Plot

It is good practice to generate a preview of the plotted drawing before sending the drawing to the printer or plotter. Generating a preview saves time and material.

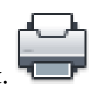
You can preview the drawing from the Plot dialog box. The preview shows exactly how the drawing will look when plotted, including lineweights, fill patterns, and other plot style options.

When you preview your drawing, the active toolbars and tool palettes are hidden and a temporary Preview toolbar is displayed that provides buttons to plot, pan, and zoom the drawing.



In the Plot and Page Setup dialog boxes, a thumbnail preview is also displayed, which shows the printable area and the position of the drawing on the page.

To preview a plot

1. Click Output tab ➤ Plot panel ➤ Plot. 
2. In the Plot dialog box, click Preview.

   A preview window opens, and the cursor changes to the real-time zoom cursor.
3. Right-click to display a shortcut menu with the following options: Plot, Pan, Zoom, Zoom Window, or Zoom Original (to zoom to the original preview magnification).
4. Press ESC to exit the preview and return to the Plot dialog box.
5. If necessary, make additional adjustments to the plot settings and preview the plotted drawing again.

6 Once the settings are correct, click OK to plot the drawing.

Toolbar: Standard 
Command entry: PLOT

Quick Reference

Commands

PAN

Moves the view in the current viewport.

PLOT

Plots a drawing to a plotter, printer, or file.

PREVIEW

Displays the drawing as it will be plotted.

ZOOM

Increases or decreases the magnification of the view in the current viewport.

System Variables

RASTERPREVIEW

Controls whether BMP preview images are saved with the drawing.

Utilities

No entries

Command Modifiers

No entries

Plot Files to Other Formats

You can export or plot your drawings in a number of formats, including DWF, DWFx, DXF, PDF, and Windows metafile (WMF). You can also output your drawings in image formats using specially designed plotter drivers.

In each case, a nonsystem plotter driver is configured to output file information. You can control the custom properties of each nonsystem driver in the Plotter Configuration Editor. Specific help for each driver is also available by choosing Help while in the custom Properties dialog box for the individual driver (accessed through the Plotter Configuration Editor).

Plot DWF Files

You can create DWF files (a 2D vector file) to publish your drawings on the web or across an intranet.

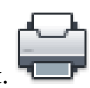
You can use the program to create DWF files. A DWF file is a 2D vector file that you can use to publish your drawing on the World Wide Web or an intranet network. Each DWF file can contain one or more drawing sheets.

DWF files can be opened, viewed, and plotted by anyone using Autodesk® Design Review. With the DWF file viewer, you can also view DWF files in Microsoft® Internet Explorer 5.01 or later. DWF files support real-time panning and zooming as well as control over the display of layers and named views.

See also:

- Publish Drawings on page 1865
- Review and Markup Files with Design Review

To plot a DWF file

1. Click Output tab ➤ Plot panel ➤ Plot. 
2. In the Plot dialog box, under Printer/Plotter, in the Name box, select the *DWF6 ePlot.pc3* configuration from the Name list.
3. Select plot settings for the DWF file as needed.
4. Click OK
5. In the Browse for Plot File dialog box, select a location and enter a file name for the DWF file.
6. Click Save.

Toolbar: Standard 
Command entry: PLOT

Quick Reference

Commands

PLOT

Plots a drawing to a plotter, printer, or file.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Plot DWFx Files

You can create DWFx files (DWF and XPS) to publish your drawings on the web or across an intranet.

You can use the DWFx ePlot (XPS Compatible) plot configuration file to plot to a DWFx file. Each DWFx file can contain one or more drawing sheets.

DWFx files can be opened, viewed, and plotted by anyone using the following applications:

- With Internet Explorer, you can view and print the 2D geometry of DWFx files.
- With Autodesk Design Review, you can view the complete DWFx file; zoom and pan within the DWF file; turn layers on and off; and mark up the drawings.

See also:

- Publish Drawings on page 1865

- Review and Markup Files with Design Review

To plot a DWFx file

1. Click Output tab ➤ Plot panel ➤ Plot.
2. In the Printer/plotter group, click the Name drop-down list, and select the *DWFx ePlot (XPS Compatible).pc3* configuration from the Name list.
3. Select plot settings for the DWFx file as needed.
4. Click OK.
5. In the Browse for Plot File dialog box, select a location and enter a file name for the DWFx file.
6. Click Save.

Toolbar: Standard
Command entry: PLOT

Quick Reference

Commands

PLOT

Plots a drawing to a plotter, printer, or file.

Plot to DXB File Formats

DXB (drawing interchange binary) file formats are supported using the DXB non-system file driver. This is commonly used to “flatten” 3D drawings to 2D.

The output is compatible with the DXBIN command and with the ADI DXB driver delivered with earlier releases. The DXB driver shares these limitations of the ADI driver:

- The driver produces 16-bit integer DXB files containing only vectors.
- DXB output is monochrome; all vectors are color 7.

■ Raster images and embedded OLE objects are not supported.

■ The driver ignores object and plot style lineweights.

See also:

■ “Configure for File Output” in the *Driver and Peripheral Guide*

To create a DXB file

1 Make sure you have configured a plotter driver for DXB file output. (See Configure for File Output in the *Driver and Peripheral Guide*.)

2 Click Output tab ➤ Plot panel ➤ Plot.

3 In the Plot dialog box, under Printer/Plotter, in the Name box, select a DXB format configuration from the list.

4 Select plot settings for the DXB file as needed.

5 Click OK.

6 In the Browse for Plot File dialog box, select a location and enter a file name for the DXB file.

7 Click Save.

Toolbar: Standard
Command entry: PLOT

Quick Reference

Commands

PLOT

Plots a drawing to a plotter, printer, or file.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Plot to Raster File Formats

The nonsystem raster driver supports several raster file formats, including Windows BMP, CALS, TIFF, PNG, TGA, PCX, and JPEG. The raster driver is most commonly used to plot to files for desktop publishing.

All but one of the formats supported by this driver produce “dimensionless” raster files that have size in pixels but do not have size in inches or millimeters. The Dimensional CALS format is for plotters that can accept CALS files. If your plotter accepts CALS files, you must specify a real paper size and resolution. Specify the resolution in dots per inch in the Vector Graphics pane of the Plotter Configuration Editor.

By default, the raster driver plots only to files. However, you can select Show All Ports on the Ports page of the Add-a-Plotter wizard or the Ports tab in the Plotter Configuration Editor; all of the ports on your computer are then available for configuration. When configured for plotting to a port, this driver plots to a file and then copies that file to the specified port. To plot successfully, make sure that the device connected to the configured port can accept and process the file. For more information, refer to the documentation provided by the device manufacturer.

The type, size, and color depth of the raster file determine the final file size. Raster files can grow very large. Use only the pixel dimensions and color depth that you need.

You can configure the background color for raster plots in the custom Properties dialog box in the Plotter Configuration Editor. If you change the background color, any objects plotted in that color are invisible.

See also:

- “Configure for File Output” in the *Driver and Peripheral Guide*

To create a raster file

1. Make sure you have configured a plotter driver for raster file output. (See Configure for File Output in the *Driver and Peripheral Guide*.)
2. Click Output tab ➤ Plot panel ➤ Plot. 
3. In the Plot dialog box, under Printer/Plotter, in the Name box, select a raster format configuration from the list.
4. Select plot settings for the raster file as needed.
5. Click OK.
6. In the Browse for Plot File dialog box, select a location and enter a file name for the raster file.
7. Click Save.

Toolbar: Standard 
Command entry: PLOT

Quick Reference

Commands

BMPOUT

Saves selected objects to a file in device-independent bitmap format.

JPGOUT

Saves selected objects to a file in JPEG file format.

PLOT

Plots a drawing to a plotter, printer, or file.

PNGOUT

Saves selected objects to a file in a Portable Network Graphics format.

TIFOUT

Saves selected objects to a file in TIFF file format.

System Variables

RASTERDPI

Controls paper size and plot scaling when changing from dimensional to dimensionless output devices, or vice versa.

Utilities

No entries

Command Modifiers

No entries

Plot Adobe PDF Files

Using the DWG to PDF driver, you can create Adobe® Portable Document Format (PDF) files from drawings.

The Adobe® Portable Document Format (PDF) is a standard for electronic information exchange. PDF files can be easily distributed for viewing and printing in the Adobe Reader available from the Adobe web site without cost. Using PDF files, you can share drawings with virtually anyone.

Like DWF6 files, PDF files are generated in a vector-based format, for maintaining precision. Drawings that are converted to PDF can be easily distributed for viewing and printing in Adobe Reader, versions 7 or later.

Use the custom Properties dialog box in the Plotter Configuration Editor to customize the output. To display this dialog box, on the Device and Document Settings tab, in the tree view, select Custom Properties. Then under Access Custom Dialog, click the Custom Properties button.

You can customize the PDF output by specifying resolution. In the Custom Properties dialog box in the Plotter Configuration Editor, you can specify the resolution for vector and raster images ranging from 150 dpi to a maximum of 4800 dpi. You can also specify custom resolutions for vector, gradient, color, and black and white output.

NOTE Although transparent objects and wipeouts are displayed correctly in the PDF viewer, they may not print with the same visual fidelity when default print settings are used. If your drawing contains transparent objects, you may need to adjust some settings in Adobe Acrobat. Set Transparency Flattening to "Print as Image" or reduce the Raster/Vector Balance in Adobe Acrobat. Refer to the Adobe documentation for more information.

See also:

- "Configure for File Output" in the *Driver and Peripheral Guide*

To plot a PDF file

1. Click Output tab ➤ Plot panel ➤ Plot.
2. In the Plot dialog box, under Printer/Plotter, in the Name box, select the *DWG to PDF.pc3* configuration from the Name list.
3. Select plot settings for the PDF file as needed.
4. Click OK.
5. In the Browse for Plot File dialog box, select a location and enter a file name for the PDF file.
6. Click Save.

Toolbar: Standard
Command entry: PLOT

To plot a PDF file in landscape orientation

1. Click Output tab ➤ Plot panel ➤ Plot.
2. In the Plot dialog box, under Printer/Plotter, in the Name box, select the *DWG to PDF.pc3* configuration from the Name list.
3. Under Paper Size, select a paper size that has the longer dimension listed first. For example, *ANSI A (11.00 x 8.50 Inches)*.
4. Click OK.
5. In the Browse for Plot File dialog box, select a location and enter a file name for the PDF file.
6. Click Save.

Toolbar: Standard

Command entry: PLOT

Quick Reference

Commands

PLOT

Plots a drawing to a plotter, printer, or file.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Plot Adobe PostScript Files

Using the Adobe PostScript driver, you can use DWGs with an array of page layout programs and archive tools such as the Adobe Acrobat Portable Document Format (PDF).

You can use the non-system PostScript driver to plot drawings to PostScript printers and PostScript files. Use the PS file format for printers and the EPS file format for files. If you plot to a hardware port, PS output is automatic. If you plot to a file and plan to copy the file to a printer, configure for PS output.

Use the custom Properties dialog box in the Plotter Configuration Editor to customize the output. To display this dialog box, on the Device and Document Settings tab, in the tree view, select Custom Properties. Then under Access Custom Dialog, click the Custom Properties button.

The PostScript driver supports three types of PostScript.

- Level 1: Use for most plotters.
- Level 1.5: Use for plotters that support color images.
- Level 2: If your plotter supports Level 2 PostScript, use to produce smaller files that print more rapidly.

The Tokenize PostScript Code and Compression options in the PostScript Custom Properties dialog box reduce output file size and improve printing speed on devices that support these options. If you have problems printing, try clearing all the options. If you successfully print with no optimizations, you can try turning the options on one at a time to determine the options your printer supports.

Some desktop publishing applications only support Level 1 PostScript. If you have problems using your EPS files, try a lower PostScript level and turn off the optimizations just described.

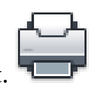
Including a preview thumbnail in your EPS file makes the file substantially larger but allows quick preview by many applications. The WMF preview is for Windows; the EPSF preview is for Macintosh and other platforms.

NOTE Including both preview images can triple your file size.

See also:

- Export PostScript Files
- “Configure for File Output” in the *Driver and Peripheral Guide*

To plot a PostScript file

1. Make sure you have configured a plotter driver for PostScript file output. (See “Configure for File Output” in the *Driver and Peripheral Guide*.)
2. Click Output tab ➤ Plot panel ➤ Plot. 
3. In the Plot dialog box, under Printer/Plotter, in the Name box, select a PostScript format configuration.
4. Select plot settings for the PostScript file as needed.
5. Click OK.
6. In the Browse for Plot File dialog box, select a location and enter a file name for the PostScript file.
7. Click Save.

Toolbar: Standard 

Command entry: PLOT

Quick Reference

Commands

PLOT

Plots a drawing to a plotter, printer, or file.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Create Plot Files

You can use any plotter configuration to create plot files that can be used with spooling software or given to service bureaus for output.

See also:

- "Set Device-Specific Configurations" in the *Driver and Peripheral Guide*

To create a plot (PLT) file

1. Click Output tab ➤ Plot panel ➤ Plot.
2. In the Plot dialog box, under Printer/Plotter, in the Name box, select a plotter configuration.

   NOTE You must use the correct plotter configuration for the output device in order to produce a valid PLT file.

   ---
3. If the Plot to File option is available and is not selected, select it.

4 Select plot settings for the plot file as needed.

5 Click OK.

6 In the Browse for Plot File dialog box, select a location and enter a file name for the plot file.

7 Click Save.

Toolbar: Standard
Command entry: PLOT

To create a batch file for plot spooling

Plot files are meant to be used with spooling software or to be given to a service bureau for output. A simple batch file can be used to output PLT files.

1 Open a text editor and save a batch file to a folder where you save your PLT files.

2 Name the batch file something like *MySpooler.bat*.

3 Add a single line to the batch file that reads:

```
copy%1 \\server\printer
```

The%1 is a parameter that the batch file will replace with the name of the *.plt* file being sent to the output device.

TIP If the output device is connected directly to the computer, you can substitute the print server path with the printer port, such as LPT1.

---

4 Save the batch file and close the text editor.

To plot a plot (PLT) file from a batch file

This procedure uses the batch file created in To create a batch file for plot spooling on page 1862.

1 Open a Command Prompt window and change directories to the folder containing your PLT file and spooling batch file.

2 At the command prompt, enter the following:

```
MySpooler.bat MyDrawing.plt
```

where *MySpooler.bat* is the name of the batch file you created and *MyDrawing.plt* is the name of the plot file.

The PLT file is copied to the output device and the drawing is created.

Quick Reference

Commands

PLOT

Plots a drawing to a plotter, printer, or file.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Publish Drawings

30

Publishing provides a streamlined alternative to plotting multiple drawings. You can easily publish an entire sheet set from the Sheet Set Manager as a set of paper drawings or as a single, electronic, multi-sheet DWF, DWFx, or PDF file.

Publishing an electronic drawing set as a DWF, DWFx, or PDF file saves time and increases productivity by providing accurate, compressed representations of drawings in a file that's easy to distribute and view.

Using Autodesk Design Review, you can view and plot DWF and DWFx files. Using Internet Explorer 7, you can view and print the 2D geometry of a DWFx file.

Overview of Publishing

An electronic drawing set is the digital equivalent of a set of plotted drawings. You create an electronic drawing set by publishing drawings to a DWF, DWFx, or PDF file.

You can publish an entire sheet set from the Sheet Set Manager. With one click, you can create an electronic drawing set by publishing the sheet set to a single, multi-sheet DWF, DWFx, or PDF file.

You can create a paper drawing set by publishing the sheet set to the plotter named in each sheet's page setup.

Using the Publish dialog box, you can assemble a collection of drawings to publish and save the list as a Drawing Set Descriptions (DSD) file. You can customize this collection of drawings for a specific user, and you can add and remove sheets as a project evolves. Once you've created a list of drawing sheets in the Publish dialog box, you can publish the drawings to any of the following:

- The plotter named in each sheet's page setup (including drawings that you want to plot to file)

- A single, multi-sheet DWF or DWFx file containing both 2D and 3D content
- A single, multi-sheet PDF file containing 2D content
- Multiple single-sheet DWF or DWFx files containing both 2D and 3D content
- Multiple single-sheet PDF files containing 2D content

Using 3D DWF publishing, you can create and publish DWF files of your three-dimensional models and view them with Autodesk Design Review.

For more on publishing 3D DWF files, refer to 3D DWF Publishing on page 1908.

Publishing for Autodesk Design Review

Publishing an electronic drawing set as a DWF or DWFx file saves time and increases productivity by providing accurate, compressed representations of drawings in a file that is easy to distribute and view. This also maintains the integrity of your original drawings.

DWF files, when published, are created in a vector-based format (except for inserted raster image content), ensuring that precision is maintained.

DWFx files are created using Microsoft's XPS format. DWFx files are ZIP files and contain metadata. This metadata can be viewed only by Autodesk Design Review.

You can view and print general graphics in Autodesk Design Review or in Internet Explorer 7. Rich DWFx metadata can only be viewed in Autodesk Design Review.

You can view or plot DWF or DWFx files using Autodesk Design Review. DWF or DWFx files can be distributed using e-mail, FTP sites, project websites, or CDs.

You can specify which block-related properties and attributes you want to make available to Autodesk Design Review users. For example, you can publish a DWF file or DWFx file for a plumbing contractor that contains block attribute information about the plumbing fixtures specified in your drawing data. And, from the same set of sheets, you can include only the block attribute data about light fixtures for an electrical contractor.

By default, jobs that are published are processed in the background, so that you can return immediately to your drawing. Only one job that you have published can be processed in the background at a time. While a job is being processed in the background, you can check its status by placing the cursor over the plotter icon on the right side of the status bar. You can also view details about all completed jobs that you have plotted or published from the current session.

See also:

- Publish, Transmit, and Archive Sheet Sets on page 510
- Specify Page Setup Settings on page 1748
- Publish 3D DWF Files on page 1908
- Review and Markup Files with Design Review

To check the status of a published job that is being processed in the background

- Place the cursor over the plotter icon in the status tray. A tooltip displays the status of the job.

To cancel part or all of a published job that is processing in the background

- Right-click the plotter icon in the status tray. Click Cancel Sheet <*sheetname*> or Cancel Entire Job.

To view details about jobs you have published

1 Do one of the following:

   - Click Output tab ➤ Plot panel ➤ View Details. 
   - In the status tray, click the plotter icon.

2 In the Plot and Publish Details dialog box, view details about published jobs.

Command entry: VIEWPLOTDETAILS
Shortcut menu: In the status tray, right-click the plotter icon. Click View Plot and Publish Details.

To turn background publishing on or off using the options dialog box

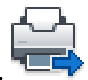
1 Click Tools menu ➤ Options.

2 In the Options dialog box, Plot and Publish tab, under Background Processing Options, select or clear the Enable Background Plot When Publishing option.

3 Click OK.

Command entry: OPTIONS

To turn background publishing on or off using the publish dialog box

1 Click Output tab ➤ Publish panel ➤ Publish. 

2 In the Publish Controls group, select or clear Publish in background.

Command entry: PUBLISH

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PUBLISH

Publishes drawings to DWF, DWFx, and PDF files, or to plotters.

System Variables

BACKGROUNDPLOT

Controls whether background plotting is turned on or off for plotting and publishing.

PUBLISHCOLLATE

Controls whether plotting a sheet set, multi-sheet plot file, or plot spool file can be interrupted by other plot jobs.

Utilities

No entries

Command Modifiers

No entries

Create and Modify a Drawing Set for Publishing

You can assemble a collection of drawing sheets to publish to a plotter, plot files, DWF, DWFx, or PDF file. You can customize your drawing set for a specific user, and you can add, remove, reorder, copy, and rename sheets in a drawing set as a project evolves.

You can publish the drawing set directly to paper and single or multiple DWF, DWFx, or PDF files. DWF, DWFx, and PDF files can be distributed using e-mail, FTP sites, project websites, or CD. You can save a description of a drawing set that has been assembled to publish in a Drawing Set Descriptions (DSD) file.

To create a drawing set for publishing

1 Open a drawing. Click Output tab ➤ Publish panel ➤ Publish. 

The Publish dialog box is displayed. If the Include Layouts When Adding Sheets option is selected, either in the shortcut menu or in the Publish dialog box, all the layouts in the current drawing are listed in the sheetlist.

2 In the Publish dialog box, you can modify the list of sheets by doing any of the following as needed:

- **Add sheets.** To add sheets from other drawings, click the Add Sheets button (or drag drawings from the desktop). In the Select Drawings dialog box, select drawings. Click Select to add them to the list of sheets in the Publish dialog box. All of the layouts in a drawing become individual sheets in the list of drawing sheets. You can remove the individual sheets if you do not want them to become part of the drawing set. A layout must be initialized (its paper size must be defined in the page setup to any size other than 0 x 0) before it can be published.

NOTE To include all layouts when you add sheets to a drawing set, ensure the Include Layouts When Adding Sheets option is selected either in the shortcut menu or in the Publish dialog box.

- **Include model layouts.** If you include an uninitialized (the paper size is not defined in the page setup or is set to 0 x 0) model layout, it will be marked as Uninitialized in the Status column on the sheet list. It can be plotted if you select an override page setup for it in the Publish dialog box from the page setup drop-down list under Page Setups in the sheet list.

NOTE To include the model when you add sheets to a drawing set, ensure the Include Model When Adding Sheets option is selected either in the shortcut menu or in the Publish dialog box.

- **Remove sheets.** To remove sheets from the list, select one or more sheets in the list. Click the Remove Sheets button. To remove all sheets, right-click the selection. Click Remove All.
- **Reorder sheets.** To reorder the sheets up or down one position in the list, select a sheet. Click either the Move Sheet Up or Move Sheet Down button. Sheets in the drawing set are viewed or plotted in the order shown in the list.
- **Rename sheets.** To rename a sheet, select it in the list and right-click. Click Rename Sheet. Enter the new sheet name.
- **Change page setups.** To change the page setup for a layout, select the sheet, and in the Page Setup list select a named page setup from the list; or select one or more sheets and right-click. Click Change Page Setup. In the Page Setup list, select a page setup, or select Import to import page setups from another drawing or template. In the Import Page Setups dialog box, select a drawing with one or more page setups. Click Import. In the Page Setup list, select a named page setup.

NOTE Change the page setup for each layout depending on your desired output. Model space page setups can only be applied to model space sheets; paper space page setups can only be applied to paper space sheets.

- **Copy sheets.** To copy one or more drawing sheets, select the sheets in the list and right-click. Click Copy Selected Sheets. The copied drawing sheets are added and highlighted at the end of the sheet list. When a sheet is copied, its name is created by the addition of *-Copy(n)* at the end of the original sheet name. For example, if you create one copy of a sheet called *Plumbing*, the copied sheet is called *Plumbing-Copy(1)*. Each time you copy the same sheet, the *n* is incremented by 1. By creating copies of a sheet, you can have different page setups and other settings for the same sheet.

3 When your list of drawing sheets is assembled and configured the way you want for your drawing set, click the Save List button.

4 In the Save List As dialog box, in the File Name box, enter a name for the list. Click Save.

   The drawing set list is saved as a DSD (Drawing Set Descriptions) file.

Toolbar: Standard
Command entry: PUBLISH

To add sheets from a drawing to a drawing set for publishing

1 Click Output tab ➤ Publish panel ➤ Publish.

2 In the Publish dialog box, click the Add Sheets button.

3 In the Select Drawings dialog box, select drawings. Click Select to add them to the list of sheets in the Publish dialog box.

   NOTE You can also drag drawings from the desktop into the Publish dialog box to add sheets to the list.

   All of the layouts in a drawing become individual sheets in the list of drawing sheets.

   You must remove the drawing sheets that you don't want to become a part of the drawing set. Layouts must be initialized before they can be published. (A layout is initialized if its paper size is defined in the page setup to any size other than 0 x 0.)

Toolbar: Standard
Command entry: PUBLISH

To add sheets from a DSD file to a drawing set for publishing

1 Click Output tab ➤ Publish panel ➤ Publish.

2 In the Publish dialog box, click the Load Sheet List button.

3 In the Load List of Sheets dialog box, select a DSD file. Click Load.

4 In the Replace or Append dialog box, click Replace to replace the current sheets with the sheets in the DSD file, or click Append to add the sheets in the DSD file to the list of sheets in the Publish dialog box.

Toolbar: Standard
Command entry: PUBLISH

To remove a sheet from a drawing set for publishing

1 Click Output tab ➤ Publish panel ➤ Publish.

2 In the Publish dialog box, ensure the sheets that you want to remove are listed.

3 Under Sheets, select one or more sheets to remove. Click the Remove Sheets button.

Toolbar: Standard
Command entry: PUBLISH

To remove all sheets from a drawing set for publishing

1 Click Output tab ➤ Publish panel ➤ Publish.

2 In the Publish dialog box, ensure the sheets that you want to remove are listed.

3 Right-click the selection. Click Remove All.

NOTE Removing all sheets cannot be undone.

Toolbar: Standard
Command entry: PUBLISH

To reorder sheets in a drawing set for publishing

1 Click Output tab ➤ Publish panel ➤ Publish.

2 In the Publish dialog box, ensure the sheets that you want to reorder are listed.

3 Select a sheet. Click the Move Sheet Up or Move Sheet Down button.

NOTE Sheets in the drawing set are viewed or plotted in the order shown in the Publish dialog box.

Toolbar: Standard
Command entry: PUBLISH

To copy sheets in a drawing set for publishing

1 Click Output tab ➤ Publish panel ➤ Publish.

2 In the Publish dialog box, ensure the sheets that you want to copy are listed.

3 Right-click the selection. Click Copy Selected Sheets.

The copied drawing sheets are added and highlighted at the end of the sheet list. When a sheet is copied, its name is created by the addition of *-copy(n)* at the end of the original sheet name.

For example, if you create one copy of a sheet called *Plumbing*, the copied sheet is called *Plumbing-Copy(1)*.

Each time you copy the same sheet, the *n* is incremented by 1. By creating copies of a sheet, you can have different page setups and other settings for the same sheet.

Toolbar: Standard
Command entry: PUBLISH

To rename sheets in a drawing set for publishing

1 Click Output tab ➤ Publish panel ➤ Publish.

2 In the Publish dialog box, ensure the sheets that you want to rename are listed.

3 Select a sheet in the Sheet list, and then do one of the following:
   - Right-click the selection. Click Rename Sheet.
   - Press F2.

4 Enter the new sheet name.

Toolbar: Standard
Command entry: PUBLISH

To change the page setup of one sheet in a drawing set for publishing

1 Click Output tab ➤ Publish panel ➤ Publish.

2 In the Publish dialog box, ensure the sheet for which you want to change the page setup is listed.

3 Select a sheet in the Sheet list.

4 Under Page Setup, select a page setup to apply to the drawing sheet.

Toolbar: Standard
Command entry: PUBLISH

To change the page setup of one or more sheets in a drawing set for publishing

1 Click Output tab ➤ Publish panel ➤ Publish.

2 In the Publish dialog box, ensure the sheets for which you want to change the page setups are listed.

3 Select one or more sheets in the list.

4 Right-click the selection. Click Change Page Setup.

5 In the Page Setup list, select a page setup to apply to the drawing sheets.

NOTE Change the page setup for each layout depending on your desired output. Model space page setups can only be applied to model space sheets; paper space page setups can only be applied to paper space sheets.

Toolbar: Standard
Command entry: PUBLISH

To import a page setup from another drawing to apply to a drawing sheet for publishing

1 Click Output tab ➤ Publish panel ➤ Publish.

2 In the Publish dialog box, ensure the sheet for which you want to change the page setup is listed.

3 Select a sheet in the Sheet list.

4 Under Page Setup, select Import from the list.

5 In the Import Page Setups dialog box, select the drawing whose page setups you want to import. Click Import.

Toolbar: Standard
Command entry: PUBLISH

To publish multiple layouts in a drawing

1 In the drawing area, click a layout tab that you want to publish.

2 Press and hold the CTRL key, and then click the other layout tabs that you want to publish.

3 Right-click one of the selected layout tabs. Click Publish Selected Layouts.

4 In the Publish dialog box, change any settings for the selected layouts.

5 Click Publish.

Command entry: PUBLISH

Quick Reference

Commands

PUBLISH

Publishes drawings to DWF, DWFx, and PDF files, or to plotters.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Create a Paper or Plot File Drawing Set

You can publish a drawing set to a plotter or plot file.

You can assemble drawing sheets into a customized drawing set and publish the sheets to the plotter named in the page setup specified for each sheet. If the plot device named in the page setup is a paper plotter, then your output will be a paper drawing set.

If the plotter is configured to plot to a file, the sheets are saved to files in the plot file location specified in the Publish Options dialog box. Each drawing sheet's plot file is saved with the same name as the sheet, with the appropriate file extension for the file (for example, *.plt*, *.jpg*, or *.bmp*). The default location can be changed in the Options dialog box, Plot and Publish tab, under Plot to File.

See also:

- Set Publish Options on page 1899

To create and publish a paper or plot file drawing set

1 Open a drawing. Click Output tab ➤ Publish panel ➤ Publish. 

2 In the Publish dialog box, the drawing layouts are shown in the Sheets List. To create the drawing set, modify the list of drawing sheets by doing any of the following as needed:

- **Add sheets.** To add sheets from other drawings, click the Add Sheets button (or drag drawings from the desktop). In the Select Drawings dialog box, select drawings. Click Select to add them to the list of sheets in the Publish dialog box. All of the layouts in a drawing become individual sheets in the list of drawing sheets. You can remove the individual sheets if you do not want them to become part of the drawing set. A layout must be initialized (its paper size must be defined in the page setup to any size other than 0 x 0) before it can be published.

NOTE To include all layouts when you add sheets to a drawing set, ensure the Include Layouts When Adding Sheets option is selected on the shortcut menu or in the Publish dialog box.

- **Include model layouts.** If an uninitialized (the paper size is not defined in the page setup or is set to 0 x 0) model layout is included, it will be marked as Uninitialized in the Status column on the sheet list. It can be plotted if a page setup override is applied.

NOTE To include the Model when you add sheets to a drawing set, ensure the Include Model When Adding Sheets option on the shortcut menu is selected.

- **Remove sheets.** To remove sheets from the list, select one or more sheets in the list. Click the Remove Sheets button. To remove all sheets, right-click. Click Remove All.

- **Reorder sheets.** To reorder the sheets up or down one position in the list, select a sheet. Click either the Move Sheet Up or Move Sheet Down button. Sheets in the drawing set are viewed or plotted in the order shown in the list.
- **Rename sheets.** To rename a sheet, select it in the list and right-click, Click Rename Sheet. Enter the new sheet name.
- **Change page setups.** To change the page setup for a layout, select the sheet. In the Page Setup list, select a named page setup from the list, or select one or more sheets in the list and right-click. Click Change Page Setup. In the Page Setup list, select a page setup or select Import to import page setups from another drawing or template. In the Import Page Setups dialog box, select a drawing with one or more page setups. Click Import. In the Page Setup list, select a named page setup.

NOTE Change the page setup for each layout depending on your desired output. Model space page setups can only be applied to model space sheets; paper space page setups can only be applied to paper space sheets.

- **Copy sheets.** To copy one or more drawing sheets, select the sheets in the list and right-click. Click Copy Selected Sheets. The copied drawing sheets are added and highlighted at the end of the sheet list. When a sheet is copied, its name is created by the addition of *-Copy(n)* at the end of the original sheet name. For example, if you create one copy of a sheet called *Plumbing*, the copied sheet is called *Plumbing-Copy(1)*. Each time you copy the same sheet, the *n* is incremented by 1. By creating copies of a sheet, you can have different page setups and other settings for the same sheet.

3. When your list of drawing sheets is assembled and configured the way you want for your paper or plot file drawing set, click the Save Sheet List button.
4. In the Save List As dialog box, in the File Name box, enter a name for the drawing set list. Click Save.

   The drawing set list is saved as a DSD (Drawing Set Descriptions) file.
5. In the Publish dialog box, under Publish To, select Plotter Named in Page Setup.
6. Click Publish to start the process.

If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress: paper drawings are being plotted or plot files are being created.

7 To view information about the processed publish job, right-click the plotter icon on the right side of the status bar. Click View Plot and Publish Details.

Toolbar: Standard 
Command entry: PUBLISH

To publish multiple layouts

1 Hold the Shift key down and click to select the layout tabs.

2 Right click and select **Publish Selected Layouts**.

3 In the Publish dialog box, select Plotter, PDF, DWF, or DWFx from the **Publish To:** drop-down list and click **Publish**.

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PLOT

Plots a drawing to a plotter, printer, or file.

PLOTTERMANAGER

Displays the Plotter Manager, where you can add or edit a plotter configuration.

PUBLISH

Publishes drawings to DWF, DWFx, and PDF files, or to plotters.

System Variables

PUBLISHCOLLATE

Controls whether plotting a sheet set, multi-sheet plot file, or plot spool file can be interrupted by other plot jobs.

Utilities

No entries

Command Modifiers

No entries

Publish an Electronic Drawing Set

You can publish an electronic drawing set as a DWF, DWFx, or PDF file.

You can assemble drawing sheets into a customized electronic drawing set. An electronic drawing set is the digital equivalent of a set of plotted drawings. This enables you to share your work with customers, suppliers, or people within your own company who may need the drawings for review or for their records.

An electronic drawing set can be saved as

- A single, multi-sheet DWF, DWFx, or PDF file
- Multiple, single-sheet DWF or DWFx, or PDF files

You can also send the published electronic drawing set as an e-mail attachment, share it using a project collaboration site such as Autodesk Buzzsaw, or post it to a website. Using Autodesk Design Review, you can view or plot only the layouts you need.

You can use the default plotter driver as installed, or you can modify configuration settings such as color depth, display resolution, file compression, font handling, and other options. Once you modify the original plotter configuration file, all future plotting and publishing of DWF, DWFx, or PDF files will be affected.

IMPORTANT Create a copy of the original plotter configuration file before you make any changes.

See also:

- Set Publish Options on page 1899
- Review and Markup Files with Design Review

To create a DWF, DWFx, or PDF file using publish

1. Open a drawing. Click Output tab ➤ Publish panel ➤ Publish. 

   The Publish dialog box is displayed. If the Include Layouts When Adding Sheets option is selected either in the shortcut menu or in the Publish dialog box, all the layouts in the current drawing are listed in the sheet list.

2. In the Publish dialog box, you can modify the list of sheets by doing any of the following as needed:

   - **Add sheets.** To add sheets from other drawings, click the Add Sheets button (or drag drawings from the desktop). In the Select Drawings dialog box, select drawings. Click Select to add them to the list of sheets in the Publish dialog box. All of the layouts in a drawing become individual sheets in the list of drawing sheets. You must remove those sheets that you don't want to become a part of the drawing set.

   NOTE To include all layouts when you add sheets to a drawing set, select — Include Layouts When Adding Sheets option in the shortcut menu or Layout Tab in the Include when adding sheets group.

   - **Include model layouts.** If you include an uninitialized (the paper size is not defined in the page setup or is set to 0 x 0) model layout, it will be marked as Uninitialized in the Status column on the sheet list. It can be plotted if you select an override page setup for it in the Publish dialog box from the page setup drop-down list under Page Setups in the sheet list.

   NOTE To include the model space when you add sheets to a drawing set, select — Include Model When Adding Sheets option in the shortcut menu or Model Tab in the Include when adding sheets group.

   - **Remove sheets.** To remove sheets from the list, select one or more sheets, and then click the Remove Sheets button. To remove all sheets, right-click. Click Remove All.

- **Reorder sheets.** To reorder the sheets up or down one position in the list, select a sheet. Click either the Move Sheet Up or Move Sheet Down button. Sheets in the drawing set are viewed or plotted in the order shown in the list.
- **Rename sheets.** To rename a sheet, select it in the list and right-click. Click Rename Sheet. Enter the new sheet name.
- **Change page setups.** To change the page setup for a layout, select the sheet, and in the Page Setup list select a named page setup from the list; or select one or more sheets and right-click. Click Change Page Setup. In the Page Setup list, select a page setup, or select Import to import page setups from another drawing or template. In the Import Page Setups dialog box, select a drawing with one or more page setups. Click Import. In the Page Setup list, select a named page setup.

NOTE Change the page setup for each layout depending on your desired output. Model space page setups can only be applied to model space sheets; paper space page setups can only be applied to paper space sheets.

- **Copy sheets.** To copy one or more drawing sheets, select the sheets in the list and right-click. Click Copy Selected Sheets. The copied drawing sheets are added and highlighted at the end of the sheet list. When a sheet is copied, its name is created by the addition of *-Copy(n)* at the end of the original sheet name. For example, if you create one copy of a sheet called *Plumbing*, the copied sheet is called *Plumbing-Copy(1)*. Each time you copy the same sheet, the *n* is incremented by 1. By creating copies of a sheet, you can have different page setups and other settings for the same sheet.

3 When your list of drawing sheets is assembled and configured the way you want for your drawing set, click the Save Sheet List button.

NOTE Ensure that you have saved your drawing before you click the Save Sheet List button.

4 In the Save List As dialog box, in the File Name box, enter a name for the list. Click Save.

The drawing set list is saved as a drawing set descriptions (DSD) file.

5 In the Publish dialog box, under Publish To, click DWF Format, and select DWF File or DWFx File. Click Publish.

6 In the Specify DWF File dialog box, enter a file name.

The Files of Type is DWF or DWFx based on the selected DWF format.

7 Click Select to provide the name and destination for the DWF or DWFx file.

NOTE You can also enter a URL so that the DWF or DWFx file is uploaded to an FTP or HTTP site.

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8 Click Save to start the electronic drawing set creation.

If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress.

9 To view information about the processed publish job, right-click the plotter icon on the right side of the status bar. Click View Plot and Publish Details.

The information in the Plot and Publish Details dialog box is also saved to the plot and Publish log file.

10 If you have the appropriate viewer installed, you can view the DWF or DWFx file. Right-click the plotter icon on the right side of the status. Click View DWF File.

Toolbar: Standard 
Command entry: PUBLISH

To create a DWF, DWFx, or PDF file using Windows Explorer

1 Launch Windows Explorer.

2 Select the drawings to publish to DWF or DWFx.

Use Shift or Ctrl + Click to select contiguous or non-contiguous files.

NOTE You can publish only 2D DWF or 2D DWFx files using the shortcut menu in Windows Explorer.

---

3 Right-click the selection. Click Publish DWF.

The Specify DWF File dialog box appears in a temporary AutoCAD session.

4 Specify Files of type as either *.dwfx or *.dwf.

5 Type a file name, or select a file.

AutoCAD publishes the drawing file with the following options:

DWF type = Multi-sheet

Password = Disabled

Layer information = Don’t include

Block information = Don’t include

By default, the DWF/ DWFx/PDF file will be saved to the same location as the selected drawing file/s.

To create and email a DWF, DWFx, or PDF file using Windows Explorer

1. Launch Windows Explorer.
2. Select the drawings to publish to DWF / DWFx.
   Use Shift or Ctrl + Click to select contiguous or non-contiguous files.

   NOTE You can publish only 2D DWF or 2D DWFx using the shortcut menu in Windows Explorer.

   ---
3. Right-click the selection. Click Publish DWF and Email.
   The Specify DWF File dialog box is displayed in a temporary AutoCAD session.
4. Specify Files of type as either *.dwfx or *.dwf.
5. Type a file name, or select a file.
   The DWF or DWFx file contains the following settings:

   DWF type = Multi-sheet

   Password = Disabled

   Layer information = Don’t include

   Block information = Don’t include
6. AutoCAD launches your default mail application with the newly created DWF or DWFx as an attachment.

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PUBLISH

Publishes drawings to DWF, DWFx, and PDF files, or to plotters.

System Variables

PUBLISHCOLLATE

Controls whether plotting a sheet set, multi-sheet plot file, or plot spool file can be interrupted by other plot jobs.

PUBLISHHATCH

Controls whether hatch patterns published to DWF or DWFx format are treated as a single object when they are opened in Autodesk Impression.

Utilities

No entries

Command Modifiers

No entries

Publish a Sheet Set

From the Sheet Set Manager, you can easily publish an entire sheet set, a subset of a sheet set, or a single sheet. It is quicker to publish a sheet set in the Sheet Set Manager rather than using the Publish dialog box.

When you publish from the Sheet Set Manager, you can publish an electronic sheet set by publishing to a DWF, DWFx, or PDF file, or you can publish a paper set by publishing to the plotter named in the page setup that is associated with each drawing sheet. You can also publish your sheets using a page setup that is saved in the page setup overrides DWT file associated with the sheet set. This page setup overrides the current page setup settings for the individual publish job.

When you open the Publish dialog box from the Sheet Set Manager, the Publish dialog box automatically lists the sheets you selected in the sheet set. You can then modify the sheet set for publishing.

NOTE You can specify that sheets are sent to the plotter in reverse order. This option is available from the Publish dialog box and from the Sheet Set Manager.

See also:

- Use Named Page Setups with Sheet Sets on page 1780
- Work with Sheets in a Sheet Set on page 479
- Set Publish Options on page 1899

To publish a sheet set to a DWF file

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager. 

2 In the Sheet Set Manager, under Sheets, select a sheet set, subset, or sheet.

3 In the upper-right corner of the Sheet Set Manager, click the Publish button.

NOTE Sheet sets cannot contain 3D DWF (or 3D DWFx) entries.

4 Click Publish to DWF from the flyout menu.

If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress. Right-click this icon for options to view the DWF file or to view information about the publish job.

TIP Publishing performance can be improved by disabling background publishing. Ensure that background publishing is not selected in the Plot and Publish tab (Tools ➤ Options), and in the Publish Controls group.

Toolbar: Standard 

Command entry: SHEETSET
Shortcut menu: Right-click a sheet set, subset, or sheet. Click Publish menu ➤ Publish To DWF.

To publish a sheet set to a DWFx file

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager.

2 In the Sheet Set Manager, under Sheets, select a sheet set, subset, or sheet.

3 In the upper-right corner of the Sheet Set Manager, click the Publish button.

> **NOTE** Sheet sets cannot contain 3D DWF (or 3D DWFx) entries.

4 Click Publish to DWFx from the flyout menu.

If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress. Right-click this icon for options to view the DWF file or to view information about the publish job.

> **TIP** Publishing performance can be improved by disabling background publishing. Ensure that background publishing is not selected in the Plot and Publish tab (Tools ➤ Options), and in the Publish Controls group.

Toolbar: Standard
Command entry: SHEETSET
Shortcut menu: Right-click a sheet set, subset, or sheet. Click Publish menu ➤ Publish To DWFx.

To publish a sheetset to a PDF file

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager.

2 In the Sheet Set Manager, right-click a sheet set, subset, or sheet. Click Publish ➤ Manage Page Setups.

3 In the Page Setup Manager dialog box, click New.

4 In the New Page Setup dialog box, under New Page Setup Name, enter a name for the PDF page setup. Click OK.

5 In the Page Setup dialog box, under Printer/Plotter, select the *DWG to PDF.pc3* configuration file.

   If the Paper Size Not Found dialog box displays, choose the paper size option for the sheet set.

6 In the Page Setup dialog box, set the desired page setup options. Click OK.

7 In the Page Setup Manager, click Close.

8 In the Sheet Set Manager, right-click the sheet set to publish. Click Publish ➤ Publish Using Page Setup Override. Click the page setup you created.

   The PDF file is plotted to the directory listed in the Plot and Publish Details (VIEWPLOTDETAILS) dialog box.

TIP Publishing performance can be improved by disabling background publishing. Background publishing can be disabled by clearing the Publish in background check box (Publish Controls group) in the Publish dialog box.

Toolbar: Standard
Command entry: SHEETSET

To publish a subset of a sheet set or an individual sheet to a DWF file

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager.

2 In the Sheet Set Manager, under Sheets, select a sheet set, subset, or sheet.

3 In the upper-right corner of the Sheet Set Manager, click the Publish button.

4 Click Publish to DWF from the flyout menu.

NOTE If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress. Right-click this icon for options to view the DWF file or to view information about the publish job.

Toolbar: Standard 
Command entry: SHEETSET
Shortcut menu: Right-click the sheet set, subset, or sheet. Click Publish ➤ Publish To DWF.

To publish a subset of a sheet set or an individual sheet to a DWFx file

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager. 

2 In the Sheet Set Manager, under Sheets, select a sheet set, subset, or sheet.

3 In the upper-right corner of the Sheet Set Manager, click the Publish button.

4 Click Publish to DWFx from the flyout menu.

NOTE If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress. Right-click this icon for options to view the DWFx file or to view information about the publish job.

Toolbar: Standard 
Command entry: SHEETSET

Shortcut menu: Right-click the sheet set, subset, or sheet. Click Publish ➤ Publish To DWFx.

To publish a subset of a sheet set or an individual sheet to a PDF file

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager. 

2 In the Sheet Set Manager, under Sheets, select a sheet set, subset, or sheet.

3 In the upper-right corner of the Sheet Set Manager, click the Publish button.

4 Click Publish to DWFx from the flyout menu.

NOTE If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress. Right-click this icon for options to view the DWFx file or to view information about the publish job.

Toolbar: Standard 
Command entry: SHEETSET
Shortcut menu: Right-click the sheet set, subset, or sheet. Click Publish ➤ Publish To DWFx.

To publish sheets in a sheet set to the plotter named in the sheet's page setup

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager. 

2 In the Sheet Set Manager, under Sheets, select a sheet set, subset, or sheet.

3 In the upper-right corner of the Sheet Set Manager, click the Publish button. Click Publish To Plotter.

NOTE If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress. Right-click this icon for options to view the DWF file or to view information about the publish job.

Toolbar: Standard 
Command entry: SHEETSET
Shortcut menu: Right-click the sheet set, subset, or sheet.Click Publish ➤ Publish To Plotter.

To publish sheets in a sheet set using a page setup from the page setup overrides file

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager. 

2 In the Sheet Set Manager, under Sheets, select the sheet set, subset, or sheet.

3 In the upper-right corner of the Sheet Set Manager, click the Publish button. Click Publish Using Page Setup Override ➤ *<filename.dwt>*. (All the named page setups that are saved in the page setup override DWT file are listed.)

The settings in the selected page setup override file take precedence over (override) the page setup settings currently set for the selected sheets. The page setup override settings apply only for this individual publish job.

NOTE If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress. Right-click this icon for options to view the DWF file or to view information about the publish job.

Toolbar: Standard 

Command entry: SHEETSET
Shortcut menu: Right-click the sheet set, subset, or sheet. Click Publish ➤ Publish Using Page Setup Override ➤ *<filename.dwt>*.

To include a plot stamp on published sheets from a sheet set

1. Click View tab ➤ Palettes panel ➤ Sheet Set Manager. 
2. In the upper-right corner of the Sheet Set Manager, click the Publish button. Click Include Plot Stamp.

   A plot stamp will be included on all published sheets.

NOTE To change the plot stamp settings, click the Publish button. Click Plot Stamp Settings.

Toolbar: Standard 
Command entry: SHEETSET
Shortcut menu: Right-click the node or name of a sheet set, subset, or an individual sheet.Click Publish ➤ Include Plot Stamp.

To publish sheets in a sheet set in reverse order

1. Click View tab ➤ Palettes panel ➤ Sheet Set Manager. 
2. In the Sheet Set Manager, under Sheets, select the name of a sheet set or a subset.
3. In the upper-right corner of the Sheet Set Manager, click the Publish button. Click Publish In Reverse Order.

   A tick mark against the menu item indicates that when you click Publish, the sheets will be published in reverse order.

NOTE This option is not available for DWF files.

Toolbar: Standard 
Command entry: SHEETSET
Shortcut menu: Right-click the node or name of a sheet set, or subset. Click Publish ➤ Publish In Reverse Order.

To set options for publishing sheet sets

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager. 

2 In the upper-right corner of the Sheet Set Manager, click the Publish button. Click Sheet Set Publish Options.

3 In the Sheet Set Publish Options dialog box, modify the settings as required.

4 Click OK.

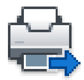
Toolbar: Standard 
Command entry: SHEETSET
Shortcut menu: Right-click the node or name of a sheet set, subset, or an individual sheet.Click Publish ➤ Sheet ions.

To modify the sheet set for publishing using the publish dialog box

1 Click View tab ➤ Palettes panel ➤ Sheet Set Manager. 

2 In the Sheet Set Manager, under Sheets, select the node or name of a sheet set, subset, or an individual sheet.

3 In the upper-right corner of the Sheet Set Manager, click the Publish button. Click Publish Dialog Box.



The Publish dialog box is displayed and lists the sheets you selected in the Sheet Set Manager.

4 In the Publish dialog box, modify the sheet list and settings as required.

5 Click Publish to publish the modified sheet set.

NOTE If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress. Right-click this icon for options to view the DWF or DWFx file or to view information about the publish job.

Toolbar: Standard 
Command entry: SHEETSET
Shortcut menu: Right-click the node or name of a sheet set, subset, or an individual sheet.Click Publish ➤ Publish Dialog Box.

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

PLOT

Plots a drawing to a plotter, printer, or file.

PUBLISH

Publishes drawings to DWF, DWFx, and PDF files, or to plotters.

SHEETSET

Opens the Sheet Set Manager.

System Variables

PUBLISHCOLLATE

Controls whether plotting a sheet set, multi-sheet plot file, or plot spool file can be interrupted by other plot jobs.

Utilities

No entries

Command Modifiers

No entries

Republish a Drawing Set

You can easily republish lists of drawing sheets that you previously saved in the Drawing Set Descriptions (DSD) file format.

After drawings have been updated, you may want to republish a drawing set for viewing or plotting. You can easily republish a sheet set (DST file) or a collection of drawing sheets that you saved as a DSD (Drawing Set Descriptions) file. You can also load and republish BP3 (Batch Plot) files.

To republish a drawing set

1. Click Output tab ➤ Publish panel ➤ Publish. 

   If you have a drawing open, all the layouts in your drawing are shown in the sheet list in the Publish dialog box. Right-click in the sheet list. Click Remove All to delete the layouts from the list of drawing sheets.

2. In the Publish dialog box, click the Load Sheet List button.

3. In the Load List of Sheets dialog box, select the location of the DSD file or the BP3 file. Click Load.

   The drawing sheets in the saved drawing set are displayed in the sheet list in the Publish dialog box.

4. In the Publish to group, do one of the following:

   - To publish to a DWF file —click DWF format, and select DWF File.
   - To publish to a DWFx file — click DWF format, and select DWFx File.

- To publish to a PDF file — click PDF format, and select PDF File.
- To publish to a plotter or printer — click Plotter Named in Page Setup.

5 Click Publish.

If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress.

6 If you have background publishing enabled, you can view information about the processed publish job. Right-click the plotter icon on the right side of the status bar. Click View Plot and Publish Details. The information in the Plot and Publish Details dialog box is also saved to the plot and publish log file.

Toolbar: Standard 
Command entry: PUBLISH

Quick Reference

Commands

PUBLISH

Publishes drawings to DWF, DWFx, and PDF files, or to plotters.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

View Electronic Drawing Sets with Autodesk Design Review

A published DWF or DWFx electronic drawing set can be viewed and plotted with Autodesk Design Review.

A published drawing set is the digital equivalent of paper plots created from your original drawings. The drawing set, which can be saved as a DWF or DWFx file, can be viewed or plotted by anyone using Autodesk Design Review.

NOTE For DWFx files, you can view the 2D geometry of a DWFx file using Internet Explorer 7.

With Autodesk Design Review, you can open, view, and print all DWF or DWFx file formats and other raster format images. You can pan, zoom, and view individual drawing sheets and viewports. You can also view layer information, sheet and sheet set properties, block information and attributes, and custom properties, if they are included in the DWF or DWFx file. As you move your cursor across DWF or DWFx geometry in the viewer, objects with associated data are displayed with a red highlighting effect. For more information about Autodesk Design Review, see Review and Markup Files with Design Review.

Recipients of drawing sets in DWF or DWFx format do not have to own or know the program. From anywhere in the world, they can view and print high-quality layouts using Autodesk Design Review.

Autodesk Design Review runs as a stand-alone application or embedded in any application that supports ActiveX controls, such as Microsoft® Internet Explorer.

For product information and a download link for the Autodesk Design Review, refer to the Products page on the Autodesk website.

See also:

- Review and Markup Files with Design Review

To view a 2D DWFx in Internet Explorer 7

- Click and drag the DWFx file to the Internet Explorer 7 window.
- Click OK at the Internet Explorer dialog box.
  A new window displays the contents of the DWFx file.

  NOTE If you have a pop-up blocker, select “Allow Blocked Content”, and click Yes when prompted.

To view the most recently published DWF file

- In the program, right-click the plotter icon on the right side of the status bar. Click View DWF file.

NOTE This option is available if you have the appropriate viewer installed on your machine.

The DWF File is displayed in Autodesk Design Review (if installed), or in the installed DWF Viewer (if you do not have Autodesk Design Review).

To view the most recently published DWFx file

- In the program, right-click the plotter icon on the right side of the status bar. Click View DWF file.
  This option is available if you have the appropriate viewer installed on your machine.

  The DWFx file is displayed in either of the following applications (if installed): Autodesk Design Review (by default), or in Internet Explorer 7 (2D DWFx files only).

Quick Reference

Commands

PUBLISH

Publishes drawings to DWF, DWFx, and PDF files, or to plotters.

System Variables

No entries

Utilities

No entries

Command Modifiers

No entries

Set Publish Options

You can set options for publishing, such as output file location, multi-sheet name options, file security (password protection), and whether or not to include layer information. You can also decide what types of information to reveal in your published DWF, DWFx, and PDF files and whether to publish drawings automatically.

You can include the following types of metadata:

- Sheet set properties (must publish using the Sheet Set Manager)
- Sheet properties (must publish using the Sheet Set Manager)
- Block standard properties and block custom properties and attributes
- Properties contained in custom objects

You use a block template (BLK) file to determine which blocks and properties to include in your published DWF, DWFx, or PDF file. You use the Block Template dialog box to create or modify the settings of a block template (BLK) file. You can also use BLK files created with the Attribute Extraction wizard.

When you change settings in the Publish Options dialog box, you can save the settings to the Drawing Set Descriptions (DSD) file to reuse the next time you publish drawings.

You can also specify whether a DWF, DWFx, or PDF file is created automatically when a drawing is saved or closed in the Auto Publish Options Dialog Box.

To change the default output location for published DWF, DWFx, PDF, and plot files

1 Click Output tab ➤ Publish panel ➤ Publish.

The Publish dialog box appears.

2 In the Publish To group, do one of the following:

- Click to select DWF Format; and select DWF File or DWFx File.

- Click to select Plotter Named in Page Setup.

3 Click Publish Options.

The Publish Options dialog box appears.

4 In the Default Output Location (DWF, DWFx, or Plot-to-File), do either of the following:

- Click Location and make a selection from the drop-down list.
- Click the [...] button, and in the Select a Folder for Generated Files dialog box, select a folder. Click Select.

5 In the Publish Options dialog box, click OK.

Toolbar: Standard
Command entry: PUBLISH

To specify the file output type to single-sheet or a single multi-sheet DWF, DWFx, or PDF file

1 Click Output tab ➤ Publish panel ➤ Publish.

The Publish dialog box appears.

2 Click Publish Options.

The Publish Options dialog box appears.

3 In the General DWF Options pane, under DWF Type, make one of the following selections:

- **Single-sheet DWF.** Specifies that an individual single-sheet DWF or DWFx file is created for each sheet.

- **Multi-sheet DWF.** Specifies that one multi-sheet DWF or DWFx file is created.

4 Click OK.

5 In the Publish dialog box, contine with publishing tasks, and then close the dialog box.

Toolbar: Standard
Command entry: PUBLISH

To specify a name for multi-sheet DWF DWFx, or PDF file

1 Click Output tab ➤ Publish panel ➤ Publish.

The Publish dialog box appears.

2 Click Publish Options.

The Publish Options dialog box appears.

3 In the General DWF Options pane, under DWF Type, select Multi-sheet DWF from the drop-down list.

4 In the Multi-sheet DWF Options pane, under DWF Naming, select Specify Name from the drop-down list.

5 Click OK.

6 In the Publish dialog box, contine with publishing tasks, and then close the dialog box.

Toolbar: Standard
Command entry: PUBLISH

To be prompted for a name for a multi-sheet DWF, DWFx, or PDF file

1 Click Output tab ➤ Publish panel ➤ Publish.

The Publish dialog box Appears.

2 Click Publish Options.

   The Publish Options dialog box appears.

3 In the General DWF Options pane, under DWF Type, select Multi-sheet DWF from the drop-down list.

4 In the Multi-sheet DWF Options, select Prompt for Name to be prompted for a file name every time you publish a DWF file.

5 Click OK.

6 In the Publish dialog box, contine with publishing tasks, and then close the dialog box.

Toolbar: Standard

Command entry: PUBLISH

To specify that DWF, DWFx, or PDF files are published with password security

1 Click Output tab ➤ Publish panel ➤ Publish.

   The Publish dialog box appears.

2 Click Publish Options.

   The Publish Options dialog box appears.

3 In the General DWF Options pane, under Password Protection, select one of the following from the drop-down list:

   - Select Specify Password, and under Password, enter a password to use for the DWF or DWFx file.
   - Select Prompt for Password. You will be prompted for a password or phrase to open the published DWF or DWFx file.

   DWF or DWFx passwords are case-sensitive. The password or phrase can consist of letters, numbers, punctuation marks, or non-ASCII characters.

   IMPORTANT Keep a list of passwords and their corresponding DWF or DWFx file names in a safe place. If you lose or forget the password, it cannot be recovered.

4 Click OK.

5 In the Publish dialog box, contine with publishing tasks, and then close the dialog box.

Toolbar: Standard
Command entry: PUBLISH

To include layer information in a published DWF, DWFx, or PDF file

1 Click Output tab ➤ Publish panel ➤ Publish.

The Publish dialog box appears.

2 Click Publish Options.

The Publish Options dialog box appears.

3 In the DWF Data Options, under Layer Information, click to display the drop-down list, and select Include.

NOTE By default, Layer Information is set to Don't Include, to reduce publishing time. If you change the setting to include layer information, you can turn individual layers on and off when you view or print the DWF file.

---

4 Click OK.

5 In the Publish dialog box, contine with publishing tasks, and then close the dialog box.

Toolbar: Standard
Command entry: PUBLISH

To include block information in a published DWF, DWFx, or PDF file

1 Click Output tab ➤ Publish panel ➤ Publish.

The Publish dialog box appears.

2 Click Publish Options.

The Publish Options dialog box appears.

3 In the DWF Data Options pane, under Block Information, click to display the drop-down list, and select Include.

NOTE By default, Block Information is set to Don't Include. If you change the setting to include block information, you can use the viewer to view or print block property and attribute information in the DWF or DWFx file.

---

4 Click OK.

5 In the Publish dialog box, contine with publishing tasks, and then close the dialog box.

Toolbar: Standard 
Command entry: PUBLISH

To include block template information in a published DWF, DWFx, or PDF file

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NOTE You can use block template (BLK) files created with the Publish Options dialog box or BLK files created using the Attribute Extraction wizard.

---

1 Click Output tab ➤ Publish panel ➤ Publish. 
The Publish dialog box appears.

2 Click Publish Options.
The Publish Options dialog box appears.

3 In the DWF Data Options pane, under Block Information, click to display the drop-down list. Select Include.

4 In the Publish Options dialog box, under DWF Data Options, Block Template File, click to display the drop-down list. Select the block template (BLK) file that contains the properties and attribute definitions you want to make available in your published DWF or DWFx file. Click OK.

NOTE The list also contains the Create and Edit options. Create opens the Publish Block Template dialog box where you can create a new block template file. Edit opens the Edit Publish Template dialog box where you can select an existing block template file to modify.

---

5 Click OK.

6 In the Publish dialog box, contine with publishing tasks, and then close the dialog box.

Toolbar: Standard 
Command entry: PUBLISH

To create a block template (BLK) file

1 Click Output tab ➤ Publish panel ➤ Publish. 

The Publish dialog box appears.

2 Click Publish Options.

The Publish Options dialog box appears.

3 In the DWF Data Options pane, under Block Information, click to display the drop-down list, and select Include.

4 Under DWF Data Options, Block Template File, click to display the drop-down list, and select Create.

The Publish Block Template dialog box opens with the Block Source Drawing area displaying the current drawing.

NOTE The working set of drawings for this list is completely independent of the drawings to include in the Publish operation.

---

5 Do one of the following:

- If this drawing contains the block properties you want to include in the published DWF or DWFx file, click Scan for Blocks.
  The program scans the DWG file for all unique block definitions and their associated properties and attributes.

- If this drawing does not contain the block properties you want to include in the published DWF or DWFx file, click Add.
  The Select Drawings dialog box is displayed. Select drawings to add to the block source drawings list. You can also add and remove drawings that contain target blocks whose properties you want to include in the published DWF or DWFx file. Click Scan for Blocks.

  The DWG file is scanned for all unique block definitions and their associated properties and attributes.

NOTE By default, block settings for nested blocks and blocks in xrefs are included. If you don't want to include these block settings, click Options in the Publish Block Template dialog box and clear the related options.

6 In the Publish Block Template dialog box, under Block Data to Publish, Unique Blocks From Source Drawings, Check Blocks to Publish, select the block names you want to include in the published DWF or DWFx files. You can right-click to select or clear all boxes.

7 Under Block Data to Publish, Properties of Selected Blocks, Check Properties to Publish, select the properties you would like to include in your published DWF or DWFx files. You can right-click to select or clear all boxes.

   The list displays the union of the properties of all selected blocks.

   NOTE If you select a block on the block list and clear the check marks of all its properties, only the name of the block is published to the resultant DWF or DWFx file; no property information is included.

8 Click Save to name and save the block template file. Click OK.

   The new block template file is now available under DWF Data Options in the Publish Block Template dialog box.

9 In the Publish dialog box, contine with publishing tasks, and then close the dialog box.

Toolbar: Standard 
Command entry: PUBLISH

To edit a block template (BLK) file

1 Click Output tab ➤ Publish panel ➤ Publish. 

   The Publish dialog box appears.

2 Click Publish Options.

   The Publish Options dialog box appears.

3 In the DWF Data Options pane, under Block Information, click to display the drop-down list, and select Include.

4 Under DWF Data Options, Block Template File, click to display the drop-down list, and select Edit.

5 In the Select Block Template dialog box, click the Block Template (BLK) file you want to edit, and then click Select.

NOTE The working set of drawings for this list is completely independent of the drawings to include in the Publish operation.

---

6 In the Publish Block Template dialog box, click Scan for Blocks.

The program scans the DWG files for all unique block definitions and their associated properties and attributes.

NOTE By default, block settings for nested blocks and blocks in xrefs are included. If you don't want to include these block settings, click Options in the Publish Block Template dialog box and clear the related options.

---

7 Under Block Data to Publish, Unique Blocks From Source Drawings, Check Blocks to Publish, select or clear the block names you want to include in the published DWF files.

8 Under Block Data to Publish, Properties of Selected Blocks, Check Properties to Publish, select or clear the properties you want to include in your published DWF or DWFx files.

This list displays the union of the properties of all selected blocks.

NOTE If you select a block from the block list and clear the check marks of all its properties, only the name of the block is published to the resultant DWF or DWFx file; no property information is included.

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9 Click Save. Click OK.

The modified block template file is now available under DWF Data Options in the Publish Block Template dialog box.

10 In the Publish dialog box, contine with publishing tasks, and then close the dialog box.

Toolbar: Standard 
Command entry: PUBLISH

Quick Reference

Commands

AUTOPUBLISH

Publishes drawings to DWF, DWFx, or PDF files automatically to a specified location.

PUBLISH

Publishes drawings to DWF, DWFx, and PDF files, or to plotters.

System Variables

AUTODWFPUBLISH

Controls whether DWF (Design Web Format) files are created automatically when you save or close drawing (DWG) files.

PUBLISHCOLLATE

Controls whether plotting a sheet set, multi-sheet plot file, or plot spool file can be interrupted by other plot jobs.

PUBLISHHATCH

Controls whether hatch patterns published to DWF or DWFx format are treated as a single object when they are opened in Autodesk Impression.

Utilities

No entries

Command Modifiers

No entries

Publish 3D DWF Files

You can create and publish DWF or DWFx files of your 3D models and view them with Autodesk Design Review.

You can generate DWF or DWFx files of your 3D models with nearly the same visual fidelity as your original DWG files. You can create a single or multi-sheet DWF or DWFx file with 2D and 3D model space objects. To access single sheet 3D DWF publishing, use the 3DDWF, EXPORT. To access multi-sheet publishing with 2D and 3D model space objects use the PUBLISH command.

Recipients of 3D DWF or 3D DWFx files can view and print them using Autodesk Design Review.

For product information and a download link for the Autodesk Design Review, refer to the Products page on the Autodesk website.

For more information about using the Autodesk Design Review, see the Autodesk Design Review Help system.

Publish Single and Multiple 3D DWF or 3D DWFx Files

To publish a single 3D DWF or 3D DWFx file, use the EXPORT or 3DDWF commands. With either of those commands running, the model you are currently working on is saved as a 3D DWF or 3D DWFx file.

To publish multiple 3D DWF or 3D DWFx files at one time, you use the PUBLISH command. The Publish dialog box displays a list of all the sheet names of the drawing files that are currently open. You can determine which sheets are published to a 3D DWF or 3D DWFx file by adding or removing sheets on the Sheet Name list.

Publishing Materials

If you assigned texture mapped materials to your models, those materials can be published with the 3D DWF or 3D DWFx file. Texture map orientation and scaling that is set in the drawing editor is retained in the published 3D DWF or 3D DWFx file.

There are some limitations to materials publishing.

- The Diffuse Map channel is the only mapping that gets published. If you use Opacity, Reflection, or Bump maps in your material, they do not get published.
- Procedural materials such as Wood or Marble do not get published.

NOTE Because the DWF Viewer and the rendering engine are different, you may encounter some deviation in how the texture maps appear in the DWF Viewer.

Increase the Smoothness of 3D DWF or 3D DWFx Models

The smoothness of 3D DWF or 3D DWFx models is improved by changing the value of the 3DDWFPREC system variable. The setting for 3DDWFPREC ranges from 1 to 6. Higher settings greatly improve the appearance of objects in the DWF Viewer.

NOTE 3DDWFPREC is a global setting that affects all objects in the 3D model. Therefore, higher values can result in very large file sizes.

Improve Performance with Large Models

Every object in your model gets processed when a DWF or DWFx file is published. When working with large models, you can greatly improve publishing performance by using blocks. For instance, your office model contains a basic cubicle arrangement of eight partition walls, a desk, and a chair. Those ten objects do not take much time to process by themselves. However, if that basic cubicle arrangement is used 100 times, it means that 1000 objects must be processed and the publishing time increases accordingly. If the basic cubicle arrangement is inserted into the model as a single block, those ten objects are processed once for each insertion.

You can improve publishing performance even more by grouping miscellaneous objects into temporary blocks. Once the publishing is complete, you can explode those blocks and continue working on those objects.

As a design gets closer to completion, you can create blocks comprised of portions of the model that are less likely to need further editing. If the first floor of the office model is not going to change much, create a block containing all of its components. During publishing, that block is processed as one object instead of each individual object on that floor.

3D DWF or 3D DWFx Unsupported Content

When publishing a 3D DWF or 3D DWFx file, some drawing content may not appear in the DWF Viewer. The content listed in the following table is not supported when you output a 3D DWF or 3D DWFx file.

Unsupported Content	Details
Animations and Walkthroughs	
Block attributes	
Font types (Various)	See the “Supported Text Fonts” table
Gradient fills (Hatches)	
Hidden edges	
Hyperlinks	

Unsupported Content	Details
Images	
Layer information	
Lights and Shadows	
Material components	■ Bump, Opacity, and Specular texture maps ■ Procedural materials (for example, wood & marble) ■ Reflection and refraction ■ Self-Illumination ■ Shininess ■ Translucency
Mtext (partial)	Bold and Italic text
Named views and cameras	
OLE objects	
Rays and X-lines	
Section clipping and XClipping	
Text thickness	
Visual styles	

Supported Text Fonts

Many text fonts are not supported when you publish a 3D DWF or 3D DWFx file. The following table lists the fonts that are published.

Supported Text Fonts
Arial
Arial Black

Supported Text Fonts
Comic Sans MS
Courier New
Impact
Lucinda Console
Lucinda Sans Unicode
Martlett
Tahoma
Times New Roman
Verdana
Verdana Italic
Webdings
Wingdings

See also:

- Review and Markup Files with Design Review

To publish a single 3D DWF file

1. Click Output tab ➤ Export to DWF/PDF panel ➤ 3D DWF. 
2. In the Export Data dialog box, specify a name and location of the DWF file. Click Save.

   By default, all model space objects are published to the 3D DWF file and, if your drawing contains xrefs, the Group By Xref Hierarchy option is active.

3 (Optional) Click Yes to open the Autodesk Design Review and view the published 3D DWF file.

Toolbar: Standard
Command entry: 3DDWF

To publish a single 3D DWFx file

1 Click Output tab ➤ Export to DWF/PDF panel ➤ 3D DWF.

2 In the Export Data dialog box, specify a name and location of the DWFx file.

3 In the Files of type drop-down list box, click 3D DWFx.

4 Click Save.

By default, all model space objects are published to the 3D DWFx file, and if your drawing contains xrefs, the Group By Xref Hierarchy option is active.

5 (Optional) Click Yes to open the Autodesk Design Review and view the published 3D DWFx file.

Toolbar: Standard

To export a 3D DWF file

1 Click Output tab ➤ Export to DWF/PDF panel ➤ 3D DWF.

2 In the Export 3D DWF dialog box, specify the name and location of the DWF file.

3 If necessary, select 3D DWF (*.dwf) from the Files Of Type list. Click Save.

By default, all model space objects are exported to the 3D DWF file and if your drawing contains xrefs, the Group By Xref Hierarchy option is active.

4 (Optional) Click Yes to open the Autodesk Design Review and view the published 3D DWF file.

Toolbar: Standard 
Command entry: EXPORT

To export a 3D DWFx file

1 Click Output tab ➤ Export to DWF/PDF panel ➤ 3D DWF. 

2 In the Export Data dialog box, specify the name and location of the DWFx file.

3 Select 3D DWFx (*.dwfx) from the Files Of Type list. Click Save.

By default, all model space objects are exported to the 3D DWFx file, and if your drawing contains xrefs, the Group By Xref Hierarchy option is active.

4 (Optional) Click Yes to open the Autodesk Design Review and view the published 3D DWFx file.

Toolbar: Standard 
Command entry: EXPORT

To publish multiple 3D or combined 2D and 3D DWF files

1 Click Output tab ➤ Plot panel ➤ Batch Plot. 

2 In the Publish dialog box, under Sheets to Publish, select the sheets you want to publish and set their Page Setup / 3D DWF setting to 3D DWF.

The sheets that are displayed in the Sheets to Publish group are controlled by the PUBLISHALLSHEETS system variable.

3 In Publish To group, click to select DWF Format.

4 Select DWF File from the drop-down list.

NOTE The Status of the selected sheet will show, “3D DWF cannot be published to a “Plotter Named in Paget Setup”, until you change the Publish to option to DWF Format.

---

5 Click Publish Options. In General DWF Options, set the DWF Type to Multi-sheet DWF.

6 Select the Publish Output options.

NOTE The Precision Override setting does not affect models sheets that are converted to 3D DWF.

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7 In 3D DWF options group, click Yes to publish materials. Click OK.

8 Click Publish.

Toolbar: Standard 
Command entry: PUBLISH

To publish multiple 3D or combined 2D and 3D DWFx files

1 Click Output tab ➤ Plot panel ➤ Batch Plot. 

2 In the Publish dialog box, under Sheets to Publish, select the sheets you want to publish and set their Page Setup / 3D DWF setting to 3D DWF.

The sheets that are displayed in the Sheets to Publish group are controlled by the PUBLISHALLSHEETS system variable.

NOTE The Status of the selected sheet will show, “3D DWF cannot be published to a “Plotter Named in Paget Setup”, until you change the Publish to option to DWF Format.

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3 In Publish To group, click to select DWF Format.

4 Select DWFx File from the drop-down list.

5 Click Publish Options. In General DWF Options, set the DWF Type to Multi-sheet DWF.

6 Select the Publish Output options.

NOTE The Precision Override setting does not affect models sheets that are converted to 3D DWF.

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7 In 3D DWF options group, click Yes to publish materials. Click OK.

8 Click Publish.

Toolbar: Standard 
Command entry: PUBLISH

To publish selected objects in your model to a 3D DWF or 3D DWFx file

1 Click Output tab ➤ Export to DWF/PDF panel ➤ 3D DWF. 

2 In the Export Data dialog box, specify a name and location for the DWF or DWFx file.

3 Click the Tools button at the upper right.

4 Click Tools menu ➤ Options.

5 In the 3D DWF Publish dialog box, Objects To Publish, click Selected Model Space Objects and then click the Select Objects button.

6 In the model, select the objects that you want to publish.

7 Press ENTER or the SPACEBAR when you're done selecting objects.

8 Click OK.

9 In the Export Data dialog box, click Save.

10 (Optional) Click Yes to open the Autodesk Design Review and view the published 3D DWF or 3D DWFx file.

Command entry: EXPORT

To view and print a published 3D DWF file or 3D DWFx file

1 If you want to view a 3D DWF or 3D DWFx file, do either of the following:
   - Immediately after the 3D DWF or 3D DWFx file has been published, click Yes to the question "Would you like to view it now?"
     Autodesk Design Review opens the 3D DWF or 3D DWFx file.
   - In Windows Explorer, double-click the 3D DWF or 3D DWFx file to open Autodesk Design Review and view the file.
   - Open Autodesk Design Review. Click File menu ➤ Open. Select the 3D DWF or 3D DWFx file you want to view.

2 If you want to print a 3D DWF or 3D DWFx file, in Autodesk Design Review, click File menu ➤ Print.
   For more information, see the Autodesk Design Review Help system.

Quick Reference

Commands

3DDWF

Creates a 3D DWF or 3D DWFx file of your 3D model and displays it in the DWF Viewer.

EXPORT

Saves the objects in a drawing to a different file format.

PUBLISH

Publishes drawings to DWF, DWFx, and PDF files, or to plotters.

System Variables

3DDWFPREC

Controls the precision of 3D DWF or 3D DWFx publishing.

PUBLISHALLSHEETS

Specifies whether to load the contents of the active document or of all open documents in the Publish dialog box.

Utilities

No entries

Command Modifiers

No entries

Print 3D Models

You can obtain physical models of 3D models created in AutoCAD.

3D Printing Service Providers

Autodesk has partnered with several 3D printing service providers. They can ouput physical models from 3D models created in AutoCAD.

When you use the 3DPRINT command, you specify 3D data from a DWG file that is then translated to a faceted mesh representation consisting of triangles. This representation is saved as binary STL file that can used by a 3D printing service provider to create a physical model.

Autodesk 3D Printing Website

To avoid errors and broken parts when you pring 3D models, it is strongly recommended that you follow guidelines on the *Autodesk 3D Printing website*. This website provides information on 3D printing service providers and *how to optimize models for 3D printing*.

How to Create and Obtain Physical Models

The following diagram illustrates, from begining to end, the basic steps of creating and obtaining a physical model:

Prepare your 3D model	To ensure successful printing of your 3D model, you may need to make some adjustments to your drawing.
Select the 3D solids you want to print	You must select the 3D solids you want to print. Only the 3D solids you select are printed, even if there are others in your drawing.
Save your drawing as an STL file	You must save your drawing as an STL file.

	3D printing service providers require that your drawing be saved in this format. After you do this, no other dialog boxes display in AutoCAD as part of the 3D printing process.
Select a service provider	You must select a 3D printing service provider. When making your decision carefully consider cost, materials, and minimum requirements. For a list of Autodesk-approved 3D printing service providers, see the Autodesk 3D Printing website.
Send the STL file to your service provider	After you select a 3D printing service provider, follow the directions on their website to obtain your model.

To send a model to a 3D printing service

1. Open the DWG file that contains the 3D model you wish to print.
2. *Optimize* your model for 3D printing.
3. Click the Output tab ➤ 3D Print panel ➤ Send to 3D Print Service.
4. In the 3D Printing - Prepare Model for Printing dialog box, click Continue.
5. In the selected DWG file, select the solids or watertight meshes you wish to print. Press ENTER. In the Send to 3D Print Service Dialog Box, under Output Preview, selected solids and watertight meshes display.
6. In the Send to 3D Print Service dialog box, under Output Dimensions, specify output dimensions, including scale and bounding box length, width, and height. Click OK.

   NOTE If you modify output dimensions, the output preview remains unchanged.

7. Save your prepared drawing as an STL file. *The Autodesk 3D Printing Website* displays.
8. Select a 3D printing service provider.

9 Follow the directions on your service provider's website to
   - Set up an account
   - Receive a price quote
   - Upload your STL file
   - Order your model
   - Pay for your model

Command entry: 3DPRINT

Quick Reference

Commands

3DPRINT

Sends a 3D model to a 3D printing service.

System Variable

SMOOTHMESHCONVERT

Sets whether mesh objects that you convert to 3D solids or surfaces are smoothed or faceted, and whether their faces are merged.

Utilities

No entries

Command Modifiers

No entries

Glossary

Commands associated with definitions are shown in parentheses at the end of the definition.

3D mesh primitive Basic mesh forms such as boxes, cones, cylinders, pyramids, wedges, spheres, and tori.

3D view Any view where the UCS icon appears in rendered colored form; current visual style is not 2D Wireframe, and the model is being viewed from an isometric view.

absolute coordinates Coordinate values measured from a coordinate system's origin point. *See also* origin, relative coordinates, user coordinate system (UCS), world coordinates, *and* world coordinate system (WCS).

acquired point In the tracking or object snap tracking methods of locating a point, an intermediate location used as a reference.

acquisition marker During tracking or object snap tracking, the temporary plus sign displayed at the location of an acquired point.

action The smallest task or user interaction that can be recorded with the Action Recorder.

Action bar Toolbar-like UI that displays the actions associated with a parameter object.

action macro A series of recorded actions that can be played back in the active drawing.

action macro file A file that stores all the actions contained in an action macro. Action macro files have the file extension *.actm*.

Action tree A control used to display the recorded actions in an action macro.

activate Part of the Autodesk software registration process. It allows you to run a product in compliance with the product's end-user license agreement.

adaptive degradation A method of controlling performance that turns off features in a certain order when performance falls below a specified level.

adaptive sampling A method to accelerate the anti-aliasing process within the bounds of the sample matrix size. *See also* anti-aliasing.

adjacent cell selection A selection of table cells that share at least one boundary with another cell in the same selection.

affine calibration A tablet calibration method that provides an arbitrary linear transformation in two-dimensional space. Affine calibration requires three calibration points to allow a tablet transformation that combines translation, independent *X* and *Y* scaling, rotation, and some skewing. Use affine calibration if a drawing has been stretched differently in the horizontal or vertical direction. (TABLET)

alias A shortcut for a command. For example, *CP* is an alias for COPY, and *Z* is an alias for ZOOM. You define aliases in the *acad.pgp* file.

aliasing The effect of discrete picture elements, or pixels, aligned as a straight or curved edge on a fixed grid appearing to be jagged or stepped. *See also* anti-aliasing.

aligned dimension A dimension that measures the distance between two points at any angle. The dimension line is parallel to the line connecting the dimension's definition points. (DIMALIGNED)

alpha channel Alpha is a type of data, found in 32-bit bitmap files, that assigns transparency to the pixels in the image.

A 24-bit truecolor file contains three channels of color information: red, green, and blue, or RGB. Each channel of a truecolor bitmap file is defined by 8 bits, providing 256 levels of intensity. The intensity of each channel determines the color of the pixel in the image. Thus, an RGB file is 24-bit with 256 levels each of red, green, and blue.

By adding a fourth, alpha channel, the file can specify the transparency, or opacity, of each of the pixels. An alpha value of 0 is transparent, an alpha value of 255 is opaque, and values in between are semi-transparent. An RGBA file (red, green, blue, alpha) is 32-bit, with the extra 8 bits of alpha providing 256 levels of transparency.

To output a rendered image with alpha, save in an alpha-compatible format such as PNG, TIFF, or Targa.

ambient color A color produced only by ambient light. Ambient color is the color of an object where it is in shadow. This color is what the object reflects when illuminated by ambient light rather than direct light.

ambient light Light that illuminates all surfaces of a model with equal intensity. Ambient light has no single source or direction and does not diminish in intensity over distance.

angular dimension A dimension that measures angles or arc segments and consists of text, extension lines, and leaders. (DIMANGULAR)

angular unit The unit of measurement for an angle. Angular units can be measured in decimal degrees, degrees/minutes/seconds, grads, and radians.

annotation scale A setting that is saved with model space, layout viewports, and model views. When you create annotative objects, they are scaled based on the current annotation scale setting and automatically displayed at the correct size.

annotational constraint Dimensional constraint used to control the size of the geometry as well as annotate the drawing.

See also parameter constraint, and dynamic constraint

annotations Text, dimensions, tolerances, symbols, notes, and other types of explanatory symbols or objects that are used to add information to your model.

annotative A property that belongs to objects that are commonly used to annotate drawings. This property allows you to automate the process of scaling annotations. Annotative objects are defined at a paper height and display in layout viewports and model space at the size determined by the annotation scale set for those spaces.

anonymous block An unnamed block created by a number of features, including associative and nonassociative dimensions.

anti-aliasing A method that reduces aliasing by shading the pixels adjacent to the main pixels that define a line or boundary. *See also* aliasing.

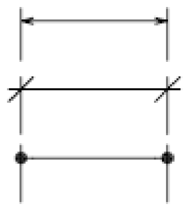
application button The button that is displayed in the top-left corner of the application. If you click the application button, the application menu is displayed.

application menu The menu that is displayed when you click the application button. The application menu contains common tools for creating, saving, and publishing a file.

approximation points Point locations that a B-spline must pass near, within a fit tolerance. *See also* fit points *and* interpolation points.

array 1. Multiple copies of selected objects in a rectangular or polar (radial) pattern. (ARRAY) 2. A collection of data items, each identified by a subscript or key, arranged so a computer can examine the collection and retrieve data with the key.

arrowhead A terminator, such as an arrowhead, slash, or dot, at the end of a dimension line showing where a dimension begins and ends.



aspect ratio Ratio of display width to height.

associative dimension A dimension that automatically adapts as the associated geometry is modified. Controlled by the DIMASSOC system variable. *See also* nonassociative dimension and exploded dimension.

associative hatch Hatching that conforms to its bounding objects such that modifying the bounding objects automatically adjusts the hatch. (BHATCH)

attenuation The diminishing of light intensity over distance.

attribute definition An object that is included in a block definition to store alphanumeric data. Attribute values can be predefined or specified when the block is inserted. Attribute data can be extracted from a drawing and inserted into external files. (ATTDEF)

attribute extraction file A text file to which extracted attribute data is written. The contents and format are determined by the attribute extraction template file. *See also* attribute extraction template file.

attribute extraction template file A text file that determines which attributes are extracted and how they are formatted when written to an attribute extraction file. *See also* attribute extraction file.

attribute prompt The text string displayed when you insert a block with an attribute whose value is undefined. *See also* attribute definition, attribute tag, *and* attribute value.

attribute tag A text string associated with an attribute that identifies a particular attribute during extraction from the drawing database. *See also attribute definition*, attribute prompt, *and* attribute value.

attribute value The alphanumeric information associated with an attribute tag. *See also attribute definition*, attribute prompt, *and attribute tag.*

AutoCAD library search path The order in which looks for a support file: current directory, drawing directory, directory specified in the support path, and directory containing the executable file, *acad.exe*.

AutoCAD window The drawing area, its surrounding menus, and the command line.

axis tripod Icon with *X*, *Y*, and *Z* coordinates that is used to visualize the viewpoint (view direction) of a drawing without displaying the drawing. (VPOINT)

B-spline curve A blended piecewise polynomial curve passing near a given set of control points. *See also* Bezier curve. (SPLINE)

back face The opposite side of a front face. Back faces are not visible in a rendered image. *See also* front faces.

base point 1. In the context of editing grips, the grip that changes to a solid color when selected to specify the focus of the subsequent editing operation. 2. A point for relative distance and angle when copying, moving, and rotating objects. 3. The insertion base point of the current drawing. (BASE) 4. The insertion base point for a block definition. (BLOCK)

baseline An imaginary line on which text characters appear to rest. Individual characters can have descenders that drop below the baseline. *See also* baseline dimension.

baseline dimension Multiple dimensions measured from the same baseline. Also called *parallel dimensions*. *See also* baseline.

basic tooltip Displays a brief description for the tooltip.

basic wheels A reference to the View Object wheel and Tour Building wheel.

Bezier curve A polynomial curve defined by a set of control points, representing an equation of an order one less than the number of points being considered. A Bezier curve is a special case of a B-spline curve. *See also* B-spline curve.

big wheels The large version of the SteeringWheels. Labels are displayed on each wheel wedge and they are larger than the size of the cursor.

bitmap The digital representation of an image having bits referenced to pixels. In color graphics, a different value represents each red, green, and blue component of a pixel.

blips Temporary screen markers displayed in the drawing area when you specify a point or select objects. (BLIPMODE)

block A generic term for one or more objects that are combined to create a single object. Commonly used for either block definition or block reference. *See also* block definition *and* block reference. (BLOCK)

block action Defines 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. A dynamic block definition usually contains at least one action that is associated with a parameter. (BACTION)

block authoring object A dimensional constraint, parameter, or action that adds intelligence to a block definition.

block authoring palettes Tool palettes used in the Block Editor to add actions and parameters to dynamic block definitions.

block authoring tools Actions, parameters, and parameter sets on the tabs of the Block Authoring Palettes window. Used in the Block Editor to create dynamic blocks.

block constraint parameter A dimensional constraint that has block authoring information associated with it.

See also: dynamic constraint

See also: annotational constraint

block definition The name, base point, and set of objects that are combined and stored in the symbol table of a drawing. *See also* block *and* block reference.

block definition table The nongraphical data area of a drawing file that stores block definitions. *See also* named object.

block instance *See* block reference.

block properties table A table that enables users to define different values for a set of properties for the block definition. Replacement for lookup properties in the future.

block reference A compound object that is inserted in a drawing and displays the data stored in a block definition. Also called *instance*. *See also* block *and* block definition. (INSERT)

bounded area A closed area that consists of a single object (such as a circle) or of multiple, coplanar objects that overlap. You can insert hatch fills within bounded areas.

Bounded areas are also used to create 3D objects through extrusion by using the PRESSPULL command.

bump map A map in which brightness values are translated into apparent changes in the height of the surface of an object.

button menu The menu for a pointing device with multiple buttons. Each button on the pointing device (except the pick button) can be defined in the customization file (*acad.cui*).

BYBLOCK A special object property used to specify that the object inherits the color or linetype of any block containing it. *See also* BYLAYER.

BYLAYER A special object property used to specify that the object inherits the color or linetype associated with its layer. *See also* BYBLOCK.

callout block A block used as symbol to reference another sheet. Callout blocks have many industry-specific terms, such as reference tags, detail keys, detail markers, and so on. *See also* label block.

camera Defines the current eye-level position in a 3D model. A camera has a location *XYZ* coordinate, a target *XYZ* coordinate, and a field of view or lens length, which determines the magnification or zoom factor.

camera target Defines the point you are viewing by specifying the coordinate at the center of the view.

candela The SI unit of luminous intensity (perceived power emitted by a light source in a particular direction) (Symbol: cd). Cd/Sr

category *See* view category.

cell The smallest available table selection.

cell boundary The four gridlines surrounding a table cell. An adjacent cell selection can be surrounded with a cell boundary.

cell style A style that contains specific formatting for table cells.

chain actions In a dynamic block definition, a property of point, linear, polar, XY, and rotation parameters. When set to Yes, a change in an action that contains the parameter in the action's selection set triggers any actions associated with that parameter, just as if you had edited the parameter in the block reference through a grip or custom property.

circular external reference An externally referenced drawing (xref) that references itself directly or indirectly. The xref that creates the circular condition is ignored.

clipping planes The boundaries that define or clip the field of view.

CMYK For *cyan, magenta, yellow, and key color*. A system of defining colors by specifying the percentages of cyan, magenta, yellow, and the key color, which is typically black.

Color bleed scale Increases or decreases the saturation of the reflected color from the material.

color map A table defining the intensity of red, green, and blue (RGB) for each displayed color.

column A vertically adjacent table cell selection spanning the height of the table. A single column is one cell in width.

command line A text area reserved for keyboard input, prompts, and messages.

compass

A visual aid that indicates the directions North, South, East, and West in the current model.

composite solid A solid created from two or more individual solids. (UNION, SUBTRACT, INTERSECT)

constraint bar Displays the geometric constraints associated with objects or with points on objects.

constraint point Point on an object that can be geometrically and/or dimensionally constrained (for example, an endpoint or an insertion point).

constraints Form of parametric design.

Rules that govern the position, slope, tangency, dimensions, and relationships among objects in a geometry.

construction plane *See* workplane.

contextual ribbon tab

A ribbon tab that is displayed only when a particular type of object, such as a hatch or table, is included in a selection. Toolbars can be changed to contextual tabs in the CUI.

continued dimension A type of linear dimension that uses the second extension line origin of a selected dimension as its first extension line origin, breaking one long dimension into shorter segments that add up to the total measurement. Also called *chain dimension*. (DIMCONTINUE)

control frame A series of point locations used as a mechanism to control the shape of a B-spline. These points are connected by a series of line segments for visual clarity and to distinguish the control frame from fit points. The SPLFRAME system variable must be turned on to display control frames.

control point *See* control frame.

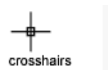
Coons patch In 3D surface meshes, the bicubic surface (one curved in the M direction and another in the N direction) interpolated between four edges.

coordinate filters Functions that extract individual *X*, *Y*, and *Z* coordinate values from different points to create a new, composite point. Also called *X,Y,Z point filters*.

crease A sharpened ridge that defines one or more edges of a mesh face subobject. (MESHCREASE)

cross sections Generally, curves or lines that define the profile (shape) of a lofted solid or surface. Cross sections can be open or closed. A lofted solid or surface is drawn in the space between the cross sections. (LOFT)

crosshairs A type of cursor consisting of two lines that intersect.



crossing selection A rectangular area drawn to select objects fully or partly within its borders.

CTB file *S*A color-dependent plot style table.

ctrl-cycle Method for cycling between different behaviors while editing geometry, either in a command or when grip-editing. Pressing and releasing the CTRL key cycles the behavior. For constrained geometry, CTRL-cycling switches between enforcing and relaxing constraints.

current drawing A drawing file that is open in the program, and receives any command or action that you enter.

cursor *See* pointer and crosshairs.

cursor menu *See* shortcut menu.

custom grips In a dynamic block reference, used to manipulate the geometry and custom properties.

custom object A type of object that is created by an ObjectARX application and that typically has more specialized capabilities than standard objects. Custom objects include parametric solids (AutoCAD Mechanical Desktop), intelligently interactive door symbols (AutoCAD Architecture), polygon objects

(AutoCAD Map 3D), and associative dimension objects (AutoCAD and AutoCAD LT). *See also* proxy object and object enabler.

customization (CUI) file An XML-based file that stores customization data. You modify a customization file through the Customize User Interface dialog box. CUI files replace MNU, MNS, and MNC files that were used to define menus in earlier releases.

data link A connection between a table and an external source of data.

decimal degrees A notation for specifying latitude and longitude. For example, 35.1234°, 100.5678°.

Latitude always precedes longitude

default drawing *See* initial environment.

default lighting The lighting in a shaded viewport when the sun and user lights are turned off. Faces are lighted by two distant light sources that follow the viewpoint as you move around the model.

default value The value that is accepted when you press ENTER at a sub-prompt. The default value is displayed in angle brackets <>. *See also* default.

definition points Points for creating a dimension. The program refers to the points to modify the appearance and value of a nonassociative dimension when the dimensioned object is modified. Also called *defpoints* and stored on the special layer DEFPOINTS.

definition table The nongraphical data area of a drawing file that stores block definitions.

dependency highlighting In a dynamic block definition, how associated objects are displayed when you select a parameter, grip, or action.

dependent named objects (in xrefs) Named objects brought into a drawing by an external reference. *See also* named object *and* symbol table.

dependent symbols *See* dependent named objects (in xrefs).

DGN underlay *See* underlay.

DIESEL For *Direct Interpretively Evaluated String Expression Language*. A macro language for altering the status line with the MODEMACRO system variable and for customizing menu items.

diffuse color An object's predominant color.

dimension line arc An arc (usually with arrows at each end) spanning the angle formed by the extension lines of an angle being measured. The dimension text near this arc sometimes divides it into two arcs. *See also* angular dimension.

dimension style A named group of dimension settings that determines the appearance of the dimension and simplifies the setting of dimension system variables. (DIMSTYLE)

dimension text The measurement value of dimensioned objects.

dimension variables A set of numeric values, text strings, and settings that control dimensioning features. (DIMSTYLE)

dimensional constraint Parametric dimensions that control the size, angle, or position of geometry relative to the drawing or other objects.When dimensions are changed, the object resizes.

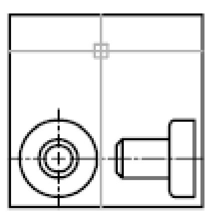
direct distance entry A method to specify a second point by first moving the cursor to indicate direction and then entering a distance.

dithering Combining color dots to give the impression of displaying more colors than are actually available.

dockable window A user interface element that can be either docked, anchored, or floating in the drawing area. Dockable windows include the command window, tool palettes, Properties Palette, and so on.

drawing area The area in which your drawings are displayed and modified. The size of the drawing area varies, depending on the size of the AutoCAD window and on how many toolbars and other elements are displayed. *See also* AutoCAD window.

drawing extents The smallest rectangle that contains all objects in a drawing, positioned on the screen to display the largest possible view of all objects. (ZOOM)



drawing extents

drawing limits *See* grid limits.

drawing set A collection of drawings assembled using the Publish dialog box.

drawing state A collection of known settings that define the behavioral properties of the drawing environment and/or drawing at a known period of time, such as when an action macro was recorded or before the playback of an action macro.

drawing template A drawing file with preestablished settings for new drawings such as *acad.dwt*and *acadiso.dwt* however, any drawing can be used as a template. *See also* initial environment.

driven constraint A non-parametric dimension enclosed in parentheses that shows the current value of geometry. The value is updated when the geometry changes size, but it does not control geometry.

driving dimension A parametric dimension that determines the size of geometry and resizes the object when its value changes.

driving property A lookup property is considered invertible when a manual change in the lookup value for a block reference causes other properties values change.

DSD For *drawing set descriptions*. A file format for saving a description of a drawing set that has been assembled using the Publish dialog box.

DST For *sheet set data*. The XML file format used to store the associations and information that define a sheet set.

DWF

An open, published, and secure file format developed by Autodesk, DWF enables you to combine and publish rich 2D- and 3D-design data and share it with others.

DWF underlay *See* underlay.

DWFx

A version of DWF based on the XML Paper Specification (XPS) from Microsoft. DWFx enables DWF files to be viewed using the free Microsoft XPS Viewer. Generically referred to as DWF.

DWG Standard file format for saving vector graphics. *See also* DWF *and* DXF.

DXF For *drawing interchange format*. An ASCII or binary file format of a drawing file for exporting drawings to other applications or for importing drawings from other applications. *See also* DWF and DWG.

dynamic constraint Dimensional constraint (Constraint Form property = "dynamic") that displays the constraints only when you select the constrained object.

See also: parameter constraint

See also: annotational constraint

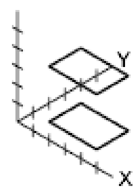
dynamic dimension Temporary dimensions that appear on objects, including dynamic block references, when they are grip edited.

edge The boundary of a face.

edge modifiers Effects such as overhang and jitter that control how edges are displayed in a shaded model.

electronic drawing set The digital equivalent of a set of plotted drawings. You create an electronic drawing set by publishing drawings to a DWF file.

elevation The default *Z* value above or below the *XY* plane of the current user coordinate system, which is used for entering coordinates and digitizing locations. (ELEV)



elevation

embed To use object linking and embedding (OLE) information from a source document in a destination document. An embedded object is a copy of the information from a source document that is placed in the destination document and has no link to the source document. *See also* link.

empty selection set A selection set that contains no objects.

enterprise customization file A CUI file that is typically controlled by a CAD manager. It is often accessed by many users and is stored in a shared network location. The file is read-only to users to prevent the data in the file from being changed. A CAD manager creates an enterprise CUI file by modifying a main CUI file and then saving the file to the support location defined in the Options dialog box, Files tab.

environment map A bitmap that is used to simulate reflections in materials that have reflective properties. The map is “wrapped” around the scene and any reflective object will show the appropriate portion of the map in the reflective parts of its material.

environment variable A setting stored in the operating system that controls the operation of a program.

expanded panel

An area on the ribbon associated with a ribbon panel. An expanded panel contains additional tools and controls. *See also* ribbon panel *and* ribbon.

explode To disassemble a complex object, such as a block, dimension, solid, or polyline, into simpler objects. In the case of a block, the block definition is unchanged. The block reference is replaced by the components of the block. *See also* block, block definition, *and* block reference. (EXPLODE)

exploded dimension Independent objects that have the appearance of a dimension but are not associated with the dimensioned object or each other. Controlled by the DIMASSOC system variable. *See also* associative dimension, nonassociative dimension, *and* explode. (EXPLODE)

extended tooltips When hovered over the tooltip for a period of time, displays additional information.

extents *See* drawing extents.

external reference (xref) A drawing file referenced by another drawing. (XREF)

extrusion A 3D solid created by sweeping an object that encloses an area along a linear path.

face A triangular or quadrilateral portion of a surface object.

face color mode A setting in the visual style that controls how color is displayed on a face.

face style A setting in the visual style that defines the shading on a face.

facet The underlying structure of the face of a 3D solid, surface, or mesh. Facets can be quadrilateral or triangular. Smoothing a mesh object increases the number of facets for each face.

feature control frame The tolerance that applies to specific features or patterns of features. Feature control frames always contain at least a geometric characteristic symbol to indicate the type of control and a tolerance value to indicate the amount of acceptable variation.

fence A multisegmented line specified to select objects it passes through.

field A specialized text object set up to display data that may change during the life cycle of the drawing. When the field is updated, the latest value of the field is displayed. (FIELD)

fill A solid color covering an area bounded by lines or curves. (FILL)

filters *See* coordinate filters.

final gathering Final gathering is an optional, additional step to calculating global illumination. Using a photon map to calculate global illumination can cause rendering artifacts such as dark corners and low-frequency variations in the lighting. You can reduce or eliminate these artifacts by turning on final gathering, which increases the number of rays used to calculate global illumination.

Final gathering can greatly increase rendering time. It is most useful for scenes with overall diffuse lighting, less useful for scenes with bright spots of indirect illumination.

You turn on final gathering on the Advanced Render Settings palette. *See also* global illumination.

First Contact balloon The interactive graphical tooltip that is displayed when the SteeringWheel is pinned during startup.

fit points Locations that a B-spline must pass through exactly or within a fit tolerance. *See also* interpolation points *and* approximation points.

fit tolerance The setting for the maximum distance that a B-spline can pass for each of the fit points that define it.

floating panel

A ribbon panel that is not attached to the rest of the ribbon or application frame.

floating viewports *See* layout viewports.

font A character set, made up of letters, numbers, punctuation marks, and symbols of a distinctive proportion and design.

footcandle The American unit of illuminance (symbol: fc). Lm/ft^2.

footcandle The American unit of illuminance (symbol: fc). Lm/ft^2

frame An individual, static image in an animated sequence. *See also* motion path.

freeze A setting that suppresses the display of objects on selected layers. Objects on frozen layers are not displayed, regenerated, or plotted. Freezing layers shortens regenerating time. *See also* thaw. (LAYER)

front faces Faces with their normals pointed outward.

general property Properties that are common between a selection of objects. These include Color, Layer, Linetype, Linetype scale, Plot style, Lineweight, Hyperlink, and Thickness.

geographic elevation The relative height along the specified up-direction defined for a geographic marker.

geographic marker Visual representation of geographic location information.

geometric constraint Rules that define the geometric relationships of objects (or points of objects) elements and control how an object can change shape or size.

Geometric constraints are coincident, collinear, concentric, equal, fix, horizontal, parallel, perpendicular, tangent, and vertical.

geometry All graphical objects such as lines, circles, arcs, polylines, and dimensions. Nongraphical objects, such as linetypes, lineweights, text styles, and layers are not considered geometry. *See also* named object.

gizmo A tool that permits you to manipulate a 3D object uniformly or along a specified axis or plane. Examples of gizmos include the 3D Move, 3D Rotate, and 3D Scale gizmos. They are displayed when you select a 3D object.

global illumination An indirect illumination technique that allows for effects such as color bleeding. As light hits a colored object in the model, photons bounce to adjacent objects and tint them with the color of the original object.

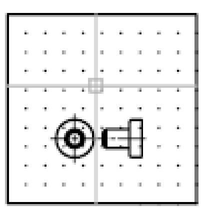
Gooch shading A type of shading that uses a transition from cool to warm colors rather than from dark to light.

graphics area *See* drawing area.

graphics window *See* AutoCAD window *and* drawing area.

grid An area covered with regularly spaced dots or lines to aid drawing. The grid spacing is adjustable. The grid dots are never plotted. *See also* grid limits. (GRID)

grid limits The user-defined rectangular boundary of the drawing area covered by dots when the grid is turned on. Also called *drawing limits*. (LIMITS)



grid limits

grip modes The editing capabilities activated when grips are displayed on an object: stretching, moving, rotating, scaling, and mirroring.

grip tool An icon that you use in a 3D view to easily constrain the movement or rotation of a selection set of objects to an axis or a plane. (3DMOVE, 3DROTATE)

grips Small squares and triangles that appear on objects you select. After selecting the grip, you edit the object by dragging it with the pointing device instead of entering commands.

ground plane The XY plane of the user coordinate system when perspective projection is turned on. The ground plane displays with a color gradient between the ground horizon (nearest to the horizon) and the ground origin (opposite the horizon). *See also* sky *and* underground.

guide curves Lines or curves that intersect each cross section of a lofted solid or surface and that define the form by adding additional wireframe information to the object. (LOFT)

handle A unique alphanumeric tag for an object in the program's database.

HDI For *Heidi Device Interface*. An interface for developing device drivers that are required for peripherals to work with the program and other Autodesk products.

heads-up display (HUD) The process of transparently displaying user interface elements on top of or over the drawing area without obscuring the view of the objects drawn on the drawing area.

helix An open 2D or 3D spiral. (HELIX)

Help menu The legacy way to access online Help. In the current version of AutoCAD, you can find Help on the InfoCenter toolbar or by pressing F1.

HLS For *hue, lightness, and saturation*. A system of defining color by specifying the amount of hue, lightness, and saturation.

Home view A special view saved with the drawing that is controlled through the ViewCube tool. The Home view is similar in concept to the default, initial view presented when a drawing is first opened.

horizontal landing An optional line segment connecting the tail of a leader line with the leader content.

horizontal ribbon

The ribbon, when it is oriented across the top of the file window.

i-drop A method by which a drawing file can be dragged from a Web page and inserted into another drawing.

IGES For *Initial Graphics Exchange Specification*. A standard format for digital representation and exchange of information between CAD/CAM systems. In AutoCAD-based products, the commands to import and export IGES files are available only in AutoCAD Mechanical.

Illuminance In photometry, illuminance is the total luminous flux incident on a surface per unit area.

indirect bump scale Scales the effect of the base material's bump mapping in areas lit by indirect light.

indirect illumination Illumination techniques such as global illumination and final gathering, that enhance the realism of a scene by simulating radiosity, or the interreflection of light between objects in a scene.

initial environment The variables and settings for new drawings as defined by the default drawing template, such as *acad.dwg* or *acadiso.dwg*. *See also* template drawing.

input property In a dynamic block definition, a parameter property other than that of a lookup, alignment, or base point parameter that you can add as a column to a lookup table. When the parameter values in a dynamic block reference match a row of input property values, the corresponding lookup property values in that table row are assigned to the block reference. (BLOOKUPTABLE)

interface element A user interface object that can be customized, such as a toolbar, pull-down menu, shortcut key, dockable window, and so on.

interpolation points Defining points that a B-spline passes through. *See also* approximation points *and* fit points.

island An enclosed area within another enclosed area. Islands may be detected as part of the process of creating hatches, polylines, and regions. (BHATCH, BOUNDARY)

ISO For *International Standards Organization*. The organization that sets international standards in all fields except electrical and electronics. Headquarters are in Geneva, Switzerland.

isometric snap style A drafting option that aligns the cursor with two of three isometric axes and displays grid, making 2D isometric drawings easier to create.

key point In a dynamic block definition, the point on a parameter that drives its associated action when edited in the block reference.

label block A block used to label views and details. Labels contain data, such as a title, view number, and scale, that is associated with the referenced view. *See also* callout block.

landing The portion of a leader object that acts as a pointer to the object being called out. A landing can either be a straight line or a spline curve.

landing gap An optional space between a leader tail and the leader content.

layer A logical grouping of data that are like transparent acetate overlays on a drawing. You can view layers individually or in combination. (LAYER)

layer index A list showing the objects on each layer. A layer index is used to locate what portion of the drawing is read when you partially open a drawing. Saving a layer index with a drawing also enhances performance when you work with external references. The INDEXCTL system variable controls whether layer and spatial indexes are saved with a drawing.

layer translation mappings Assignments of a set of layers to another set of layers that defines standards. These standards include layer names and layer properties. Also called *layer mappings*.

layout The environment in which you create and design paper space layout viewports to be plotted. Multiple layouts can be created for each drawing.

layout viewports Objects that are created in paper space that display views. *See also* paper space. (VPORTS)

leader tail The portion of a leader line that is connected to the annotation.

lens length Defines the magnification properties of a camera's lens. The greater the lens length, the narrower the field of view.

level of smoothness The property assigned to a mesh object to control how much the edges of the object are smoothed. Level 0 (zero) represents the least rounded shape for a specified mesh object. Higher levels result in increased smoothness.

light glyph The graphic representation of a point light or a spotlight.

limits *See* drawing limits.

line font *See* linetype.

linetype How a line or type of curve is displayed. For example, a continuous line has a different linetype than a dashed line. Also called *line font.* (LINETYPE)

lineweight A width value that can be assigned to all graphical objects except TrueType® fonts and raster images.

link To use object linking and embedding (OLE) to reference data in another file. When data is linked, any changes to it in the source document are automatically updated in any destination document. *See also* embed.

LL84 coordinate system Common latitude longitudinal-based coordinate system where latitude and longitude are both measured from -90 to 90 degrees.

Longitude begins at 0 degrees at the Prime Meridian in Greenwich, England and is measured from -180 to 180.

Latitude is 0 degrees at the equator and is measured from -90 to 90.

lofted solid/surface A solid or surface that is drawn through a set of two or more cross-section curves. The cross sections define the profile (shape) of the resulting solid or surface. Cross sections (generally, curves or lines) can be open or closed. (LOFT)

lookup property In a dynamic block definition, a lookup parameter that you add to a lookup table. The lookup parameter label is used as the property name. When the parameter values in a dynamic block reference match a row of input property values, the corresponding lookup property values in that table row are assigned to the block reference. (BLOOKUPTABLE)

lookup table Defines properties for and assigns property values to a dynamic block. Assigns property values to the dynamic block reference based on how the block is manipulated in a drawing. (BLOOKUPTABLE)

lumen The SI unit of luminous flux (Symbol: lm). Cd * Sr

luminaire This refers to the aggregation of a lamp or lamps and its fixture. The fixture may be a simple can or a complex armature with constrained joints.

luminance Luminance is the value of light reflected off a surface. It is a measure of how bright or dark we perceive the surface.

luminous flux The perceived power in per unit of solid angle. The total luminous flux for a lamp is the perceived power emitted in all directions.

lux The SI unit of illuminance (symbol: lx). Lm/m^2

macro-derived selection A selection set of all the objects that have been created during the playback of an action macro up to the command that is requesting a selection set.

main customization file A writable CUI file that defines most of the user interface elements (including the standard menus, toolbars, keyboard accelerators, and so on). The *acad.cui* file (the default main CUI file) is automatically loaded when you start AutoCAD.

markup A single comment or a redline geometry correction inserted into a DWF file using Autodesk Design Review.

markup set A group of markups contained within a single DWF file.

merge In tables, an adjacent cell selection that has been combined into a single cell.

mesh A tessellated, or subdivided object type that is defined by faces, edges, and vertices. Mesh can be smoothed to achieve a more rounded appearance and creased to introduce ridges. Before AutoCAD 2010, only the less modifiable polygon and polyface mesh was available.

mini wheels The small version of SteeringWheels. No labels are displayed on any of the wedges and they are often the size of the cursor.

mirror To create a new version of an existing object by reflecting it symmetrically with respect to a prescribed line or plane. (MIRROR)

mode A software setting or operating state.

model A two- or three-dimensional representation of an object.

model space One of the two primary spaces in which objects reside. Typically, a geometric model is placed in a three-dimensional coordinate space called model space. A final layout of specific views and annotations of this model is placed in paper space. *See also* paper space. (MSPACE)

model viewports A type of display that splits the drawing area into one or more adjacent rectangular viewing areas. *See also* layout viewports, TILEMODE, *and* viewport. (VPORTS)

motion path Defines the path or target of a camera. A path can be a line, arc, elliptical arc, circle, polyline, 3D polyline, or spline.

multi-sheet DWF A DWF file that contains multiple sheets.

multileader A leader object that creates annotations with multiple leader lines.

named object Describes the various types of nongraphical information, such as styles and definitions, stored with a drawing. Named objects include linetypes, layers, dimension styles, text styles, block definitions, layouts, views, and viewport configurations. Named objects are stored in definition (symbol) tables.

named objects, dependent *See* dependent named objects (in xrefs).

named path A saved motion path object that is linked to a camera or target.

named range A tool in Microsoft Excel that provides a method to assign a meaningful name to a single cell or a range of cells.

named view A view saved for restoration later. (VIEW)

node An object snap specification to locate points, dimension definition points, and dimension text origins.

nonassociative dimension A dimension that does not automatically change as the associated geometry is modified. Controlled by the DIMASSOC system variable. *See also* associative dimension *and* exploded dimension.

normal A normal is a vector that defines which way a face is pointing. The direction of the normal indicates the front, or outer surface of the face.

noun-verb selection Selecting an object first and then performing an operation on it rather than entering a command first and then selecting the object.

NURBS For *nonuniform rational B-spline curve*. A B-spline curve or surface defined by a series of weighted control points and one or more knot vectors. *See also* B-spline curve.

object One or more graphical elements, such as text, dimensions, lines, circles, or polylines, treated as a single element for creation, manipulation, and modification. Formerly called *entity*.

object enabler A tool that provides specific viewing and standard editing access to a custom object when the ObjectARX application that created the custom object is not present. *See also* custom object and proxy object.

Object Snap mode Methods for selecting commonly needed points on an object while you create or edit a drawing. *See also* running object snap *and* object snap override.

object snap override Turning off or changing a running Object Snap mode for input of a single point. *See also* Object Snap mode *and* running object snap.

ObjectARX (AutoCAD Runtime Extension) A compiled-language programming environment for developing AutoCAD applications.

OLE For *object linking and embedding*. An information-sharing method in which data from a source document can be linked to or embedded in a destination document. Selecting the data in the destination document opens the source application so that the data can be edited. *See also* embed *and* link.

opacity map Projection of opaque and transparent areas onto objects, creating the effect of a solid surface with holes or gaps.

origin The point where coordinate axes intersect. For example, the origin of a Cartesian coordinate system is where the *X*, *Y*, and *Z* axes meet at 0,0,0.

Ortho mode A setting that limits pointing device input to horizontal or vertical (relative to the current snap angle and the user coordinate system). *See also* snap angle *and* user coordinate system (UCS).

orthogonal Having perpendicular slopes or tangents at the point of intersection.

output property A lookup property whose value is determined by input properties (other parameter properties) through the use of a lookup table.

page setup A collection of plot device and other settings that affect the appearance and format of the final output. These settings can be modified and applied to other layouts.

pan To shift the view of a drawing without changing magnification. *See also* zoom. (PAN)

paper space One of two primary spaces in which objects reside. Paper space is used for creating a finished layout for printing or plotting, as opposed to doing drafting or design work. You design your model using the Model tab. *See also* model space *and* viewport. (PSPACE)

parameter In a dynamic block definition, defines custom properties for the dynamic block by specifying positions, distances, and angles for geometry in the block.

parameter set A tool on the Parameter Sets tab of the Block Authoring Palettes window that adds one or more parameters and one or more associated actions to the dynamic block definition.

parametric design Ability to establish relationships between objects, to drive the size and orientation of geometry with model and user-defined parameters.

parametric drawing Feature in AutoCAD that assigns constraints to objects, establishing the distance, location, and orientation of objects with respect to other objects.

partial customization file Any CUI file that is not defined as the main CUI file. You can load and unload partial CUI files as you need them during a drawing session.

path curve Defines the direction and length that a profile curve is lofted, swept, or extruded to create a solid or surface. (SWEEP, LOFT, EXTRUDE)

PC2 file Complete plotter configuration file. PC2 files contain all plot settings and device-specific settings that were saved in previous versions. *See also* PCP file *and* PC3 file.

PC3 file Partial plotter configuration file. PC3 files contain plot settings information such as the device driver and model, the output port to which the device is connected, and various device-specific settings, but do not include any custom plotter calibration or custom paper size information. *See also* PMP file, STB file, *and* CTB file.

PCP file Partial plotter configuration file. PCP files contain basic plot specifications and pen parameters that were saved in previous versions. Plot settings that are stored in a PCP file include pen assignments, plotting units, paper size, plot rotation, plot origin, scale factor, and pen optimization level. *See also* PC2 file *and* PC3 file.

performance tuning A method of optimizing 3D graphics performance. The performance tuner examines your graphics card and 3D display driver and decides whether to use software or hardware implementation for features that support both.

personalization Customizes the executable file during installation with the user name, company, and other information.

perspective view Objects in 3D seen by an observer positioned at the viewpoint looking at the view center. Objects appear smaller when the distance from the observer (at the view point) to the view center increases. Although a perspective view appears realistic, it does not preserve the shapes of objects. Parallel lines seemingly converge in the view. The program has perspective view settings for VPORTS table entries as well as viewport objects.

photometric lights Photometric lights are physically-correct lights. Physically correct lights attenuate as the square of the distance. Photometry is the science of measurement of visible light in terms of its perceived brightness.

photon map A technique to generate the indirect illumination effects of global illumination used by the renderer. When it calculates indirect illumination, the renderer traces photons emitted from a light. The photon is traced through the model, being reflected or transmitted by objects, until it strikes a diffuse surface. When it strikes a surface, the photon is stored in the photon map.

photorealistic rendering Rendering that resembles a photograph.

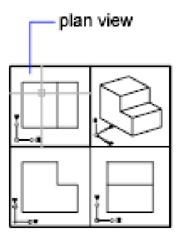
pick button The button on a pointing device that is used to select objects or specify points on the screen. For example, on a two-button mouse, it is the left button by default.

pick points Clicking and acquiring a point on an object in the drawing.

pick-first The selection of objects before an action macro or command is issued.

pick-first set A selection set of objects that are selected prior to the execution of an action macro or a command. *See also* **pre-selection set**.

plan view A view orientation from a point on the positive *Z* axis toward the origin (0,0,0). (PLAN)



planar face A flat face that can be located anywhere in 3D space.

planar projection Mapping of objects or images onto a plane.

planar surface A flat surface that can be located anywhere in 3D space. (PLANESURF)

playback The process of executing the actions stored in a previously recorded action macro.

pline *See* polyline.

plot style An object property that specifies a set of overrides for color, dithering, gray scale, pen assignments, screening, linetype, lineweight, endstyles, joinstyles, and fill styles. Plot styles are applied at plot time.

plot style table A set of plot styles. Plot styles are defined in plot style tables and apply to objects only when the plot style table is attached to a layout or viewport.

plug-ins *Plug-ins* are libraries of reuseable content that extend the functionality of AutoCAD. Plug-ins are created by third party developers and can be accessed from the Featured Technologies and Content channel of the Communications Center.

PMP file *Plot Model Parameter*. File containing custom plotter calibration and custom paper size information associated with plotter configuration file.

point 1. A location in three-dimensional space specified by *X*, *Y*, and *Z* coordinate values. 2. An object consisting of a single coordinate location. (POINT)

point filters *See* coordinate filters.

pointer A cursor on a video display screen that can be moved around to place textual or graphical information. *See also* crosshairs.

polar array Objects copied around a specified center point a specified number of times. (ARRAY)

Polar Snap A precision drawing tool used to snap to incremental distances along the polar tracking alignment path. *See also* polar tracking on page 1946.

polar tracking A precision drawing tool that displays temporary alignment paths defined by user-specified polar angles. *See also* Polar Snap.

polyface and polygon mesh Legacy mesh types that were available before AutoCAD 2010. Although you can continue to create polygonal and polyface mesh (for example, by setting MESHTYPE to 0), the newer, more modifiable mesh type is recommended.

polygon window selection A multisided area specified to select objects in groups. *See also* crossing selection *and* window selection.

polyline An object composed of one or more connected line segments or circular arcs treated as a single object. Also called *pline*. (PLINE, PEDIT)

polysolid A swept solid that is drawn the same way you draw a polyline or that is based on an existing line. By default, a polysolid always has a rectangular profile. You can specify the height and width of the profile. (POLYSOLID)

pre-playback drawing environment The drawing state prior to the playback of an action macro.

pre-selection set A selection set of objects that is defined prior to the execution of an action macro.

primary table fragment The fragment of a broken table that contains the beginning set of rows up to the first table break.

primitive Basic 3D forms such as boxes, cones, cylinders, pyramids, wedges, spheres, and tori. You can create primitive meshes and primitive 3D solid objects.

procedural materials Materials that generate a 3D pattern in two or more colors, and apply it to an object. These include marble and wood. Also called *template materials*.

profile curve An object that is swept, extruded, or revolved and defines the shape of the resulting solid or surface. (SWEEP, EXTRUDE, REVOLVE)

prompt A message on the command line or in a tooltip that asks for information or requests action such as specifying a point.

proxy object A substitute for a custom object when the ObjectARX application that created the custom object is not available. *See also* custom object and object enabler.

push pin A push pin-shaped button used on the ribbon and in the application menu. On the ribbon, push pins are used to keep a ribbon panel expanded. In the application menu, push pins keep an item in the list of recently opened items.

PWT A template file format used to publish drawings to the Web.

Quick View image A thumbnail image of a drawing, layout or model space that is displayed using Quick View tools.

Quick View tool A tool to preview and switch between open drawings and layouts in a drawing.

ray tracing The renderer can generate reflections and refractions. Ray tracing traces the path of rays sampled from the light source. Reflections and refractions generated this way are physically accurate.

You turn on ray tracing on the Advanced Render Settings palette.

ray-traced shadows A way that the renderer can generate shadows. Ray tracing traces the path of rays sampled from the light source. Shadows appear where rays have been blocked by objects. Ray-traced shadows have sharp edges.

Ray-traced shadows are active when Shadow Map is turned off on the Advanced Render Settings palette.

recorded value The input captured during the recording of an action macro for a sub-prompt of a command.

rectangular break To break a table into multiple parts that are evenly spaced and set at a user-specified height using the table breaking grips.

redraw To quickly refresh or clean up blip marks in the current viewport without updating the drawing's database. *See also* regenerate. (REDRAW)

reference A definition, known as an external reference or block reference, that is used and stored in the drawing. *See also* block (BLOCK) *and* external reference (xref). (XREF)

refine To quadruple the number of faces in a mesh object as you reset the baseline level of smoothness. (You cannot make a mesh courser than its baseline level.) You can also refine specified mesh faces without resetting the baseline level of smoothness for the object. (MESHREFINE)

reflectance scale Increases or decreases the amount of energy the material reflects.

reflection color The color of a highlight on shiny material. Also called *specular color*.

reflection line In a dynamic block reference, the axis about which a flip action's selection set flips when the associated parameter is edited through a grip or the Properties palette.

reflection mapping Creates the effect of a scene reflected on the surface of a shiny object.

refraction How light distorts through an object.

regenerate To update a drawing's screen display by recomputing the screen coordinates from the database. *See also* redraw. (REGEN)

region Two-dimensional enclosed areas that have physical properties such as centroids or centers of mass. You can create regions from objects that form closed loops. They area commonly created in order to apply hatching and shading. (REGION)

relative coordinates Coordinates specified in relation to previous coordinates.

relax constraints Ability to temporarily ignore constraints while editing geometry. After the geometry is edited, the constraints are either removed or retained based on whether the constraint is still valid for the edited geometry.

request user input An item that is assigned to an action node that pauses the playback of an action macro so a user can provide some form of input before playback resumes.

reverse lookup Adds a lookup grip to a dynamic block reference. When you click this grip, a drop-down list of the lookup values for that lookup property (column in the lookup table) is displayed. When you select a value from the list, the corresponding input property values are assigned to the block reference. Depending on how the block was defined, this usually results in a change in the block reference's geometry. (BLOOKUPTABLE)

rewind Restores the previous view or movement path created by the Autodesk® ViewCube® navigation tool, SteeringWheels, and other navigation tools.

RGB For *red, green, and blue*. A system of defining colors by specifying percentages of red, green, and blue.

ribbon A palette that displays buttons and controls used for both 2D drawing and annotation and 3D modeling, viewing, and rendering. *See also* ribbon tab *and* ribbon panel *and* slide-out panel. (RIBBON)

ribbon panel

A labeled control in the ribbon. Ribbon panels contain buttons or other controls. Multiple ribbon panels make a ribbon tab.

ribbon tab

The most general control on the ribbon. Ribbon tabs contain ribbon panels, which contain buttons or other controls.

roll arrows Curved arrows located above the ViewCube tool with which you can rotate the current view 90 degrees clockwise or counterclockwise.

roughness Value to simulate how light hitting a face is reflected back to the user. A high roughness value simulates a non-shiny or rough object (sandpaper/carpet). A low roughness value simulates a very shiny object (metals, some plastics.)

row A horizontally adjacent table cell selection spanning the width of the table. A single row is one cell in height.

RSS feed Information published by a website to which you subscribe. Usually allows users to receive notifications when new content (articles) are posted. RSS stands for Rich Site Summary (or Really Simple Syndication).

rubber-band line A line that stretches dynamically on the screen with the movement of the cursor. One endpoint of the line is attached to a point in your drawing, and the other is attached to the moving cursor.

running object snap Setting an Object Snap mode so it continues for subsequent selections. *See also* Object Snap mode *and* object snap override. (OSNAP)

sampling Sampling is an antialiasing technique. It provides a "best guess" color for each rendered pixel. The renderer first samples the scene color at locations within the pixel or along the pixel's edge, then uses a filter to combine the samples into a single pixel color.

save back To update the objects in the original reference (external or block reference) with changes made to objects in a working set during in-place reference editing.

scale representation The display of an annotative object based on the annotation scales that the object supports. For example, if an annotative object supports two annotations scales, it has two scale representations

script file A set of commands executed sequentially with a single SCRIPT command. Script files are created outside the program using a text editor, saved in text format, and stored in an external file with the file extension *.scr*.

search tag A user-defined keyword used to search for commands in the menu browser.

secondary table fragment Any fragment of a broken table that does not contain the beginning set of rows.

selection node A specific type of action tree node that is used to handle selection activities.

selection sensitivity The ability to define the pivot point for reorienting a model based on the current selection.

selection set One or more selected objects that a command can act upon at the same time.

In a dynamic block definition, the geometry associated with an action.

shadow maps A shadow map is a bitmap that the renderer generates during a pre-rendering pass of the scene. Shadow maps don't show the color cast by transparent or translucent objects. On the other hand, shadow maps can have soft-edged shadows, which ray-traced shadows cannot.

Shadow mapped shadows provide softer edges and can require less calculation time than ray-traced shadows, but are less accurate. On the Advanced Render Settings palette, shadow mapped shadows are active when Shadow Map is turned on.

ShapeManager ShapeManager is the Autodesk technology that provides 3D solid modeling to AutoCAD and other products.

sheet A layout selected from a drawing file and assigned to a sheet set. *See also* sheet set.

sheet list table A table listing all sheets in a sheet set. A sheet list table can be generated automatically with the Sheet Set Manager.

sheet selection A named selection of sheets in a sheet set that can be conveniently recalled for archiving, transmitting, and publishing operations.

sheet set An organized and named collection of sheets from several drawing files. *See also* sheet. (SHEETSET)

shortcut keys Keys and key combinations that start commands; for example, CTRL+S saves a file. The function keys (F1, F2, and so on) are also shortcut keys. Also known as *accelerator keys*.

shortcut menu The menu displayed at your cursor location when you right-click your pointing device. The shortcut menu and the options it provides depend on the pointer location and other conditions, such as whether an object is selected or a command is in progress.

shot A saved view that can later be restored by name or with ShowMotion. A shot can contain a static thumbnail of the saved view or camera motion that can be played back as an animation.

ShowMotion User interface element where you can access named views (shots) that are stored in the current drawing. The named views (shots) are organized by sequences and can contain movements.

sky The background color of the drawing area when perspective projection is turned on. The sky displays with a color gradient between the sky horizon (nearest to the horizon) and the sky zenith (opposite the horizon). *See also* ground plane.

slide file A file that contains a raster image or snapshot of the objects displayed in the drawing area. Slide files have the file extension *.sld*. (MSLIDE, VSLIDE)

slide library A collection of slide files organized for convenient retrieval and display. Slide library names have the extension *.slb* and are created with the *slidelib.exe* utility.

slide-out panel An area on the ribbon associated with a ribbon panel. A slide-out panel contains additional tools and controls. *See also* ribbon panel *and* ribbon.

smooth shading Smoothing of the edges between polygon faces.

smoothness A property of mesh objects that controls the roundness of the object. Objects with higher levels of smoothness have more faces, or tessellations.

snap angle The angle that the snap grid is rotated.

snap grid The invisible grid that locks the pointer into alignment with the grid points according to the spacing set by Snap. Snap grid does not necessarily correspond to the visible grid, which is controlled separately by GRID. (SNAP)

Snap mode A mode for locking a pointing device into alignment with an invisible rectangular grid. When Snap mode is on, the screen crosshairs and all input coordinates are snapped to the nearest point on the grid. The snap resolution defines the spacing of this grid. *See also* Object Snap mode. (SNAP)

snap resolution The spacing between points of the snap grid.

solid history A property of a solid that allows you to see and modify the original forms of the solid.

solid object An object that represents the entire volume of an object, for example a box.

solid primitive A basic solid form. Solid primitives include: box, wedge, cone, cylinder, sphere, torus, and pyramid.

spatial index A list that organizes objects based on their location in space. A spatial index is used to locate what portion of the drawing is read when you partially open a drawing. Saving a spatial index with a drawing also enhances performance when working with external references. The INDEXCTL system variable controls whether layer and spatial indexes are saved with a drawing.

specular reflection The light in a narrow cone where the angle of the incoming beam equals the angle of the reflected beam.

split face A mesh face that has been subdivided along a specified vector.

STB file For *plot style table* file. Contains plot styles and their characteristics.

SteeringWheels Tool set that provides access to 2D and 3D navigation tools.

stretch frame In a dynamic block definition that contains a stretch action or a polar stretch action, determines how the objects within or crossed by the frame are edited in the block reference.

sub-prompt A command prompt that instructs for a form of input to complete a command or alter a property.

subdivision A division, or tessellation in a mesh object. As a mesh object is smoothed, the number of subdivisions increases.

subobject Any part of a solid: a face, an edge, or a vertex. Also, an original individual form that is part of a composite solid.

Subscription Center Subscription members can access the latest releases of Autodesk software, incremental product enhancements, personalized web support, and self-paced e-Learning in InfoCenter. To access Subscription Center, go to the InfoCenter toolbar and click the Key button.

subset A named collection of sheets in a sheet set that is often organized by discipline or workflow stage. *See also* view category.

surface An open-ended, infinitely thin object that corresponds to the shape of a 3D object. You can create surfaces in several ways. For example, you can convert from other objects such as mesh, or with sweep, loft, or revolve operations that create open-ended objects.

surface normal Positive direction perpendicular to the surface of an object.

swept solid/surface A solid or surface created in the shape of the specified profile (the swept object) swept along the specified path. (SWEEP)

symbol A representation of an item commonly used in drawings. Symbols are inserted in drawings as blocks.

symbol library A collection of block definitions stored in a single drawing file.

symbol table *See* definition table *and* block definition table.

system variable A name that is recognized as a mode, size, or limit. Read-only system variables, such as DWGNAME, cannot be modified directly by the user.

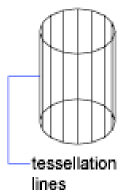
table A rectangular array of cells that contain annotation, primarily text but also blocks. In the AEC industry, tables are often referred to as “schedules” and contain information about the materials needed for the construction of the building being designed. In the manufacturing industry, they are often referred to as “BOM” (bills of materials). (TABLE)

table break The point at the bottom of a table row where the table will be split into a supplementary table fragment.

table style A style that contains a specific table format and structure. A table style contains at least 3 cell styles.

temporary files Data files created during an program session. The files are deleted by the time you end the session. If the session ends abnormally, such as during a power outage, temporary files might be left on the disk.

tessellation lines Lines that help you visualize a curved surface.



In a 3D mesh object, tessellations indicate the boundaries of the mesh faces.

text style A named, saved collection of settings that determines the appearance of text characters—for example, stretched, compressed, oblique, mirrored, or set in a vertical column.

texture map The projection of an image (such as a tile pattern) onto an object (such as a chair).

thaw A setting that displays previously frozen layers. *See also* freeze. (LAYER)

thickness The distance certain objects are extruded to give them a 3D appearance. (PROPERTIES, CHPROP, ELEV, THICKNESS)

tiled viewports *See* model viewports.

TILEMODE A system variable that controls whether viewports can be created as movable, resizable objects (layout viewports), or as nonoverlapping display elements that appear side-by-side (model viewports). *See also* viewport.

tool message

A small instructional message that appears over the drawing window and is specific to the active navigation tool from a SteeringWheel.

toolbar Part of the interface containing icons that represent commands.

tooltip A small box of text that identifies or explains an object or interface element when the cursor hovers near or over it.

tracking A way to locate a point relative to other points on the drawing.

tracking menu

A cluster of buttons that follows the cursor as you move it over the window.

translucency How light is scattered through an object.

transmittance scale Increases or decreases the amount of energy a transparent material transmits out to the scene.

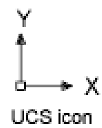
transparency A quantity defining how much light is let through an object.

transparent command A command started while another is in progress. Precede transparent commands with an apostrophe.

two sided material The positive and negative normal of the material will be considered during the rendering process.

UCS *See* user coordinate system (UCS).

UCS icon An icon that indicates the orientation of the UCS axes. (UCSICON)



underconstrained geometry Objects with unsolved degrees of freedom are underconstrained.

underground The XY plane of the user coordinate system when perspective projection is turned on and when viewed from below ground. The underground plane displays with a color gradient between the earth horizon (nearest to the horizon) and the earth azimuth (opposite the horizon). *See also* ground plane *and* sky.

underlay A DWF, or DGN file used to provide visual context in a drawing file. Underlays cannot be edited, and do not provide the full range of notification. Underlays cannot be bound to a drawing. *See also* external reference (xref).

up direction A vector defining what direction is Up. By default this is the positive Z – axis (0,0,+1).

The up direction and the north direction are always constrained such that they are perpendicular to each other.

user coordinate system (UCS) A user-defined coordinate system that defines the orientation of the *X*, *Y*, and *Z* axes in 3D space. The UCS determines the default placement of geometry in a drawing. *See also* world coordinate system (WCS).

user parameter Named user-defined variable (real number or an expression) that can be used in expressions for dimensional constraints or other user parameters.

UVW The material's coordinate space. Used instead of XYZ because that is usually reserved for the world coordinate system (WCS). Most material maps are a 2D plane assigned to a 3D surface. The U, V, and W coordinates parallel the relative directions of X, Y, and Z coordinates. If you look at a 2D map image, U is the equivalent of X, and represents the horizontal direction of the map. V is the equivalent of Y, and represents the vertical direction of the map. W is the equivalent of Z and represents a direction perpendicular to the UV plane of the map.

value node A specific type of action node that is used to handle requests for user input and hold the recorded value that was captured during the recording of an action macro.

value set In a dynamic block definition, a range or list of values specified for a linear, polar, XY, or rotation parameter.

vector A mathematical object with precise direction and length but without specific location.

vertex A location where edges or polyline segments meet.

vertical ribbon

The ribbon when it is oriented vertically, usually on the left or right of the file window.

view A graphical representation of a model from a specific location (viewpoint) in space. *See also* viewpoint *and* viewport. (3DORBIT, VPOINT, DVIEW, VIEW)

view category A named collection of views in a sheet set that is often organized by function. *See also* subset.

ViewCube User interface element that displays the current orientation of a model, and allows you to interactively rotate the current view or restore a preset view.

viewpoint The location in 3D model space from which you are viewing a model. *See also* view *and* viewport. (3DORBIT, DVIEW, VPOINT)

viewport A bounded area that displays some portion of the model space of a drawing. The TILEMODE system variable determines the type of viewport created. 1. When TILEMODE is off (0), viewports are objects that can be moved and resized on a layout. (MVIEW) 2. When TILEMODE is on (1), the entire drawing area is divided into nonoverlapping model viewports. *See also* TILEMODE, view, *and* viewpoint. (VPORTS)

viewport configuration A named collection of model viewports that can be saved and restored. (VPORTS)

virtual screen display The area in which the program can pan and zoom without regenerating the drawing.

visibility mode Displays or does not display geometry (in a dimmed state) that is invisible for a visibility state. (BVMODE)

visibility state In a dynamic block, a custom property that allows only specified geometry to display in the block reference. (BVSTATE)

visual style A collection of settings that control the display of edges and shading in a viewport.

volumetric shadows A photorealistically rendered volume of space cast by the shadow of an object.

watertight A closed 3D solid or mesh that has no gaps.

WCS *See* world coordinate system (WCS).

wheel

A reference to one of the individual user interface elements that make up SteeringWheels. See also SteeringWheels.

wheel surface

Area of a SteeringWheel that is used to organize wedges and other buttons.

wheel wedge

A section on the surface of a SteeringWheel that is designated for a specific navigation or orientation tool.

wheels

A reference to more than one of the individual user interface elements that make up SteeringWheels. See also SteeringWheels.

window selection A rectangular area specified in the drawing area to select multiple objects at the same time. *See also* crossing selection, polygon window selection.

wipeout object A polygonal area that masks underlying objects with the current background color. This area is bounded by the wipeout frame, which you can turn on for editing and turn off for plotting.

wireframe model The representation of an object using lines and curves to represent its boundaries.

working drawing A drawing for manufacturing or building purposes.

working set A group of objects selected for in-place reference editing.

workplane Another name for the XY plane of the user coordinate system. *See also* elevation *and* user coordinate system (UCS).

workspace A set of menus, toolbars and dockable windows (such as the Properties palette, DesignCenter, and the Tool palettes window) that are grouped and organized so that you can work in a custom, task-oriented drawing environment.

world coordinate system (WCS) A coordinate system used as the basis for defining all objects and other coordinate systems. *See also* user coordinate system (UCS).

world coordinates Coordinates expressed in relation to the world coordinate system (WCS).

wrap around

Behavior where the cursor wraps around the window and appears on the opposite side to allow the continuation of a drag operation instead of stopping at the edge of the drawing area.

X,Y,Z **point filters** *See* coordinate filters.

xref *See* external reference (xref).

zoom To reduce or increase the apparent magnification of the drawing area. (ZOOM)

Index

A

B

C

D

E

F

G

H

I

M

N

O

P

Q

R

S

T

U

V

W