AutoCAD 2012

User's Guide

Autodesk®

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Part 1: Get Information

Find the Information You Need

1

There are various ways to find information about how to use this program, and multiple resources are available.

This AutoCAD is often intuitive, but when you do need to look something up, you can save time and avoid frustration if you use the Help system to find information. The Help system is organized in a structured design that makes information easy to locate.

Access Help and Other Sources of Information

You can use InfoCenter to search for information, and to access Autodesk Exchange for online services and resources.

Overview of InfoCenter and Autodesk Exchange

InfoCenter provides a convenient way to search for topics in the Help system, sign in to Autodesk ID, open Autodesk Exchange, and display the options in the Help menu. It can also display product announcements, updates, and notifications.



TIP For best results when you use the Search box in InfoCenter or in Autodesk Exchange, enter at least two keywords.

Autodesk Exchange for AutoCAD

Autodesk Exchange provides a web-based experience directly within the product. Included are tabs that access the following sources of information:

- Home. Provides a wide variety of content, including announcements, expert tips, videos, and links to blogs. When enabled for online access, the Home tab also includes access to the Knowledge Base, Communication Center, and Subscription Center.
- Apps. Offers popular accessories, libraries, and plug-ins for purchase.
- Help. Opens the AutoCAD Help system.

NOTE The tabs for online sources are not available in all languages, products, and installations. Autodesk Exchange can be configured for offline use only, in which case only the Home and Help tabs are displayed.

Receive Product Updates and Announcements

Whenever new information is available, InfoCenter notifies you by displaying a balloon message. Click the link in the balloon message to open the article or announcement.

If you don't want to receive balloon notifications, in the Options dialog box, System tab, turn off Balloon Notification.

Quick Reference

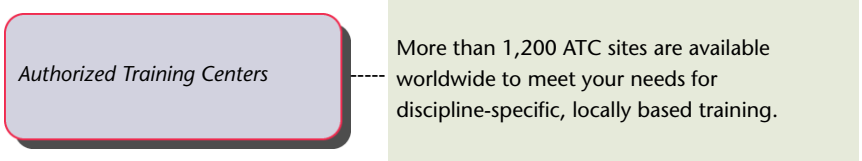
Commands

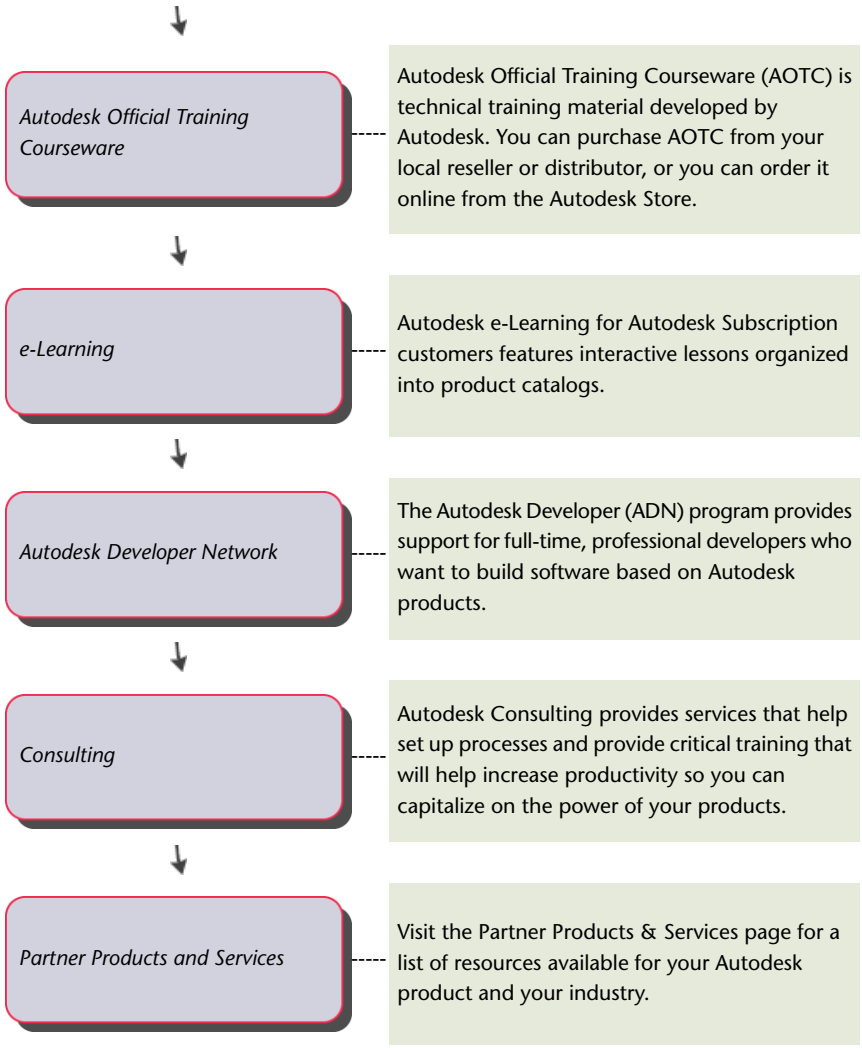
OPTIONS

Customizes the program settings.

Learn the Product

For the latest information about Autodesk training, visit *http://www.autodesk.com/training* or contact your local Autodesk office.





View the Product Readme

You can find late-breaking information about this software in the online *Readme*.

It is suggested that you read through the online *Readme* for the latest information about recommended hardware, updated installation instructions, and known software problems.

View the online readme

Where to Find the AutoCAD Express Tools

The AutoCAD Express Tools contain a library of productivity tools designed to help you extend the power of AutoCAD.

They are easy to use, easily integrated into your menus and toolbars, and cover a wide range of functions, including dimensioning, drawing, and selecting and modifying objects.

The AutoCAD Express Tools are installed as a Typical installation of AutoCAD. If you do not want them installed, you must disable them during installation by selecting the Express Tools option on the Select the Installation Type page.

For more detailed information about using Express Tools, refer to the AutoCAD Express Tools Help system found on the Express menu (in the Classic Menu Bar on page 39).

NOTE Express Tools are available in their original form, in English only, and are not supported. Double-byte characters are not supported.

To verify if Express Tools are loaded

- At the Command prompt, enter **EXPRESSTOOLS** to enable the tools. You can install the AutoCAD Express Tools as part of the AutoCAD installation, or you can add them later using Add or Remove Programs on the Control Panel.

To access the Express Tools and their toolbars

- To display the Express menu, enter **EXPRESSMENU** at the Command prompt.
- To display the Express toolbars, right-click next to any docked toolbar. Click EXPRESS, and select an Express toolbar. Repeat these steps until all of the Express toolbars you want are displayed.

  TIP With the Express toolbars displayed, select a workspace and save it. The Express Tools toolbars will then always be included with that workspace.

Join the Customer Involvement Program

If you participate in the Customer Involvement Program (CIP), specific information about how you use AutoCAD is forwarded to Autodesk. This information includes what features you use the most, problems that you encounter, and other information helpful to the future direction of the product.

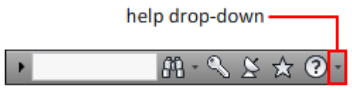
See the following links for more information.

- Learn more about the Autodesk Customer Involvement Program: *http://www.autodesk.com/cip*
- Read the Autodesk Privacy Statement: *http://www.autodesk.com/cipprivacy*

When you join, you will be able to view reports that can help you optimize your use of AutoCAD.

To turn the CIP on or off

1. On the InfoCenter toolbar, to the right of the Help button, click the drop-down arrow.
2. Click Customer Involvement Program.
3. In the Customer Involvement Program dialog box, select to start or stop participating.
4. Click OK.



Get Information from Drawings

2

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, amount of free space on your disk drive, and total amount of time spent in the drawing file.

See also:

- Enter System Variables on the Command Line on page 49
- Add Identifying Information to Drawings on page 214
- Extract Geometric Information from Objects on page 662

- Compare Dimension Styles and Variables on page 1531

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.

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.

Quick Reference

Commands

QSELECT

Creates a selection set based on filtering criteria.

Part 2: The User Interface

Tools in the Application Window

3

Use the Application menu, ribbon, and other elements in the application window to access frequently used commands, and control the operation of the product.

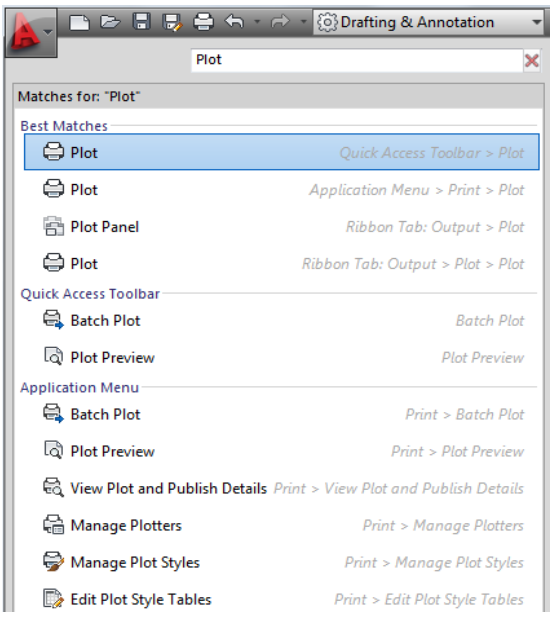
The Application Menu

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.



Quick Reference

Commands

CUI

Manages the customized user interface elements in the product.

OPTIONS

Customizes the program settings.

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.

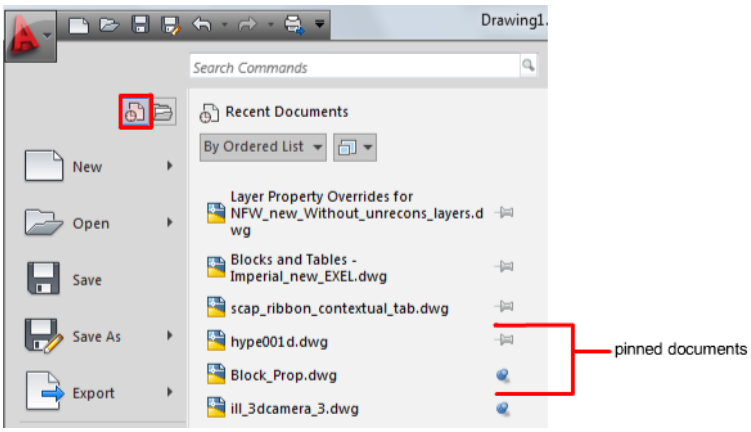
Browse Files

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.

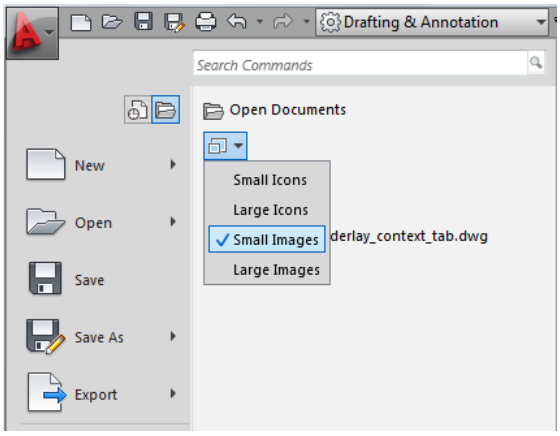
Sort and Group Options

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.

Quick Reference

Commands

CUI

Manages the customized user interface elements in the product.

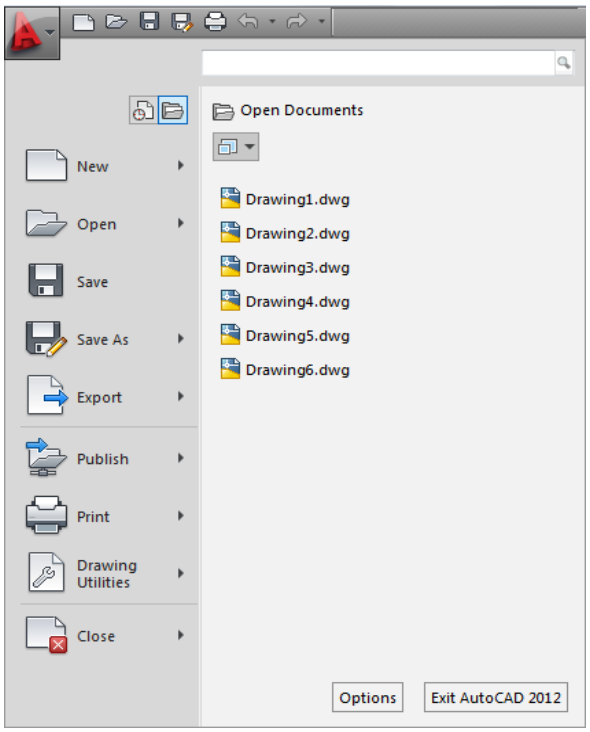
OPTIONS

Customizes the program settings.

Currently Open Documents

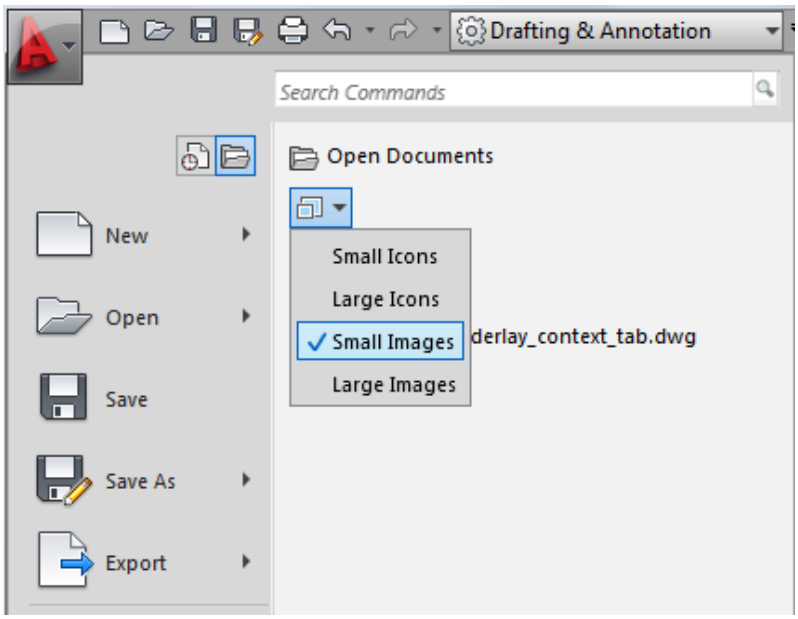
View only files that are currently open with the Open Documents list.

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.

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

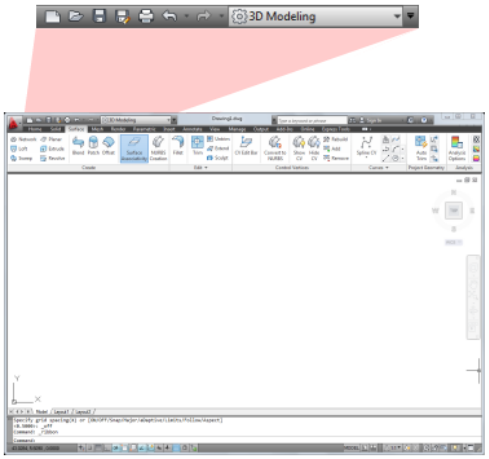
Manages the customized user interface elements in the product.

OPTIONS

Customizes the program settings.

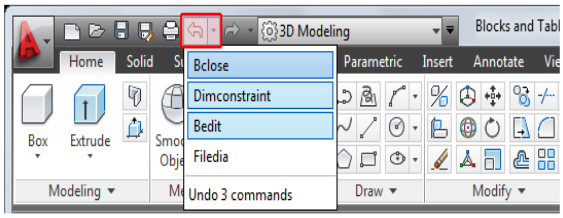
Quick Access Toolbar

Display frequently used tools with the Quick Access toolbar.



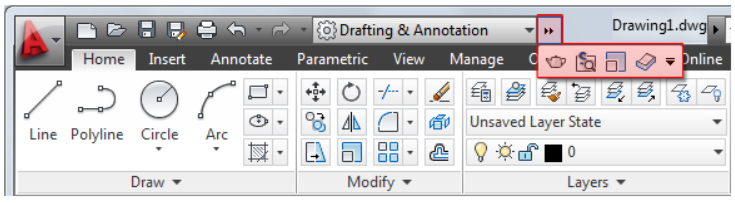
View Undo and Redo History

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

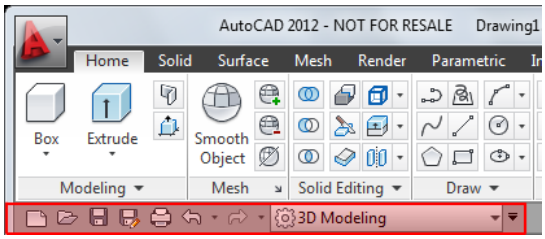
Add unlimited tools to the Quick Access toolbar. Tools that extend past the maximum length of the toolbar are displayed in a flyout button.



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 40

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.

To remove a command from the Quick Access toolbar

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

Quick Reference

Commands

CUI

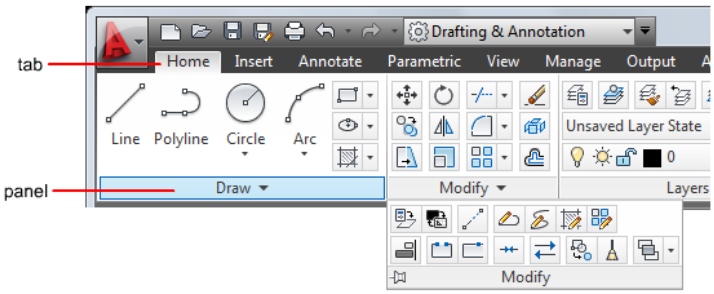
Manages the customized user interface elements in the product.

The Ribbon

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

Overview of the Ribbon

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



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

Limits the number of objects that can be changed at one time with the ribbon property controls or a contextual tab.

RIBBONDOCKEDHEIGHT

Determines whether the horizontally docked ribbon is set to the height of the current tab or a predetermined height.

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.

Display and Organize the Ribbon

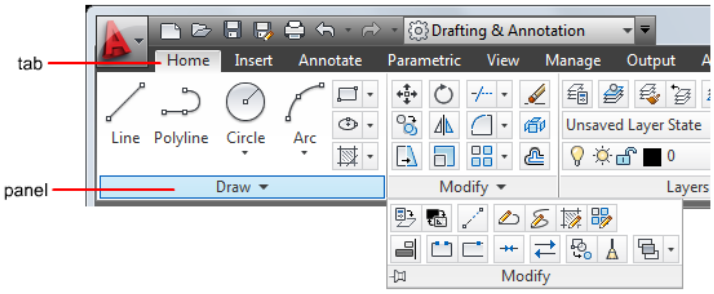
The ribbon is displayed horizontally or vertically.

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 file window.

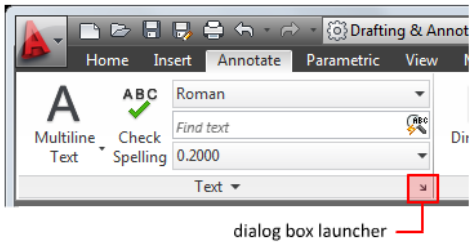
The vertical ribbon can also float in the file window or on a second monitor.

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.



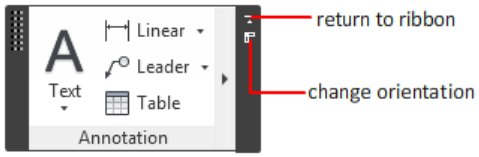
Some ribbon panels display a dialog box related to that panel. The dialog box launcher is denoted by an arrow icon, , in the lower-right corner of the panel. The dialog box launcher indicates that you can display a related dialog box. Display the related dialog box by clicking the dialog box launcher.



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

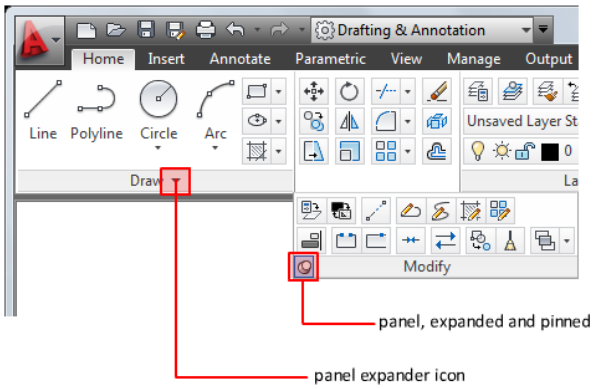
If you pull a panel off of a ribbon tab and into the drawing area or onto another monitor, that panel floats where you placed it. The floating panel remains open until you return it to the ribbon, even if you switch ribbon tabs.



Slideout Panels

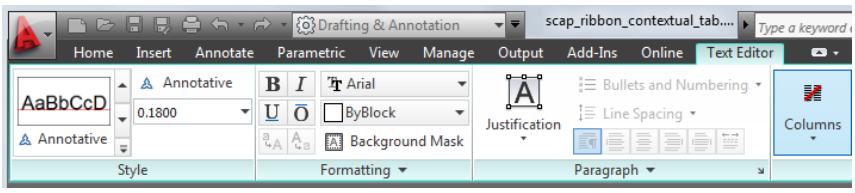
An arrow in the middle of a panel title, , indicates that you can slide out the panel to display additional tools and controls. Click on the title bar of an open panel to display the slideout panel. By default, a slideout panel

automatically closes when you click another panel. To keep a panel expanded, click the push pin, , in the bottom-left corner of the slideout panel.



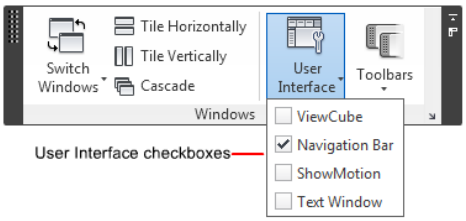
Contextual Ribbon Tabs

When you select a particular type of object or 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.



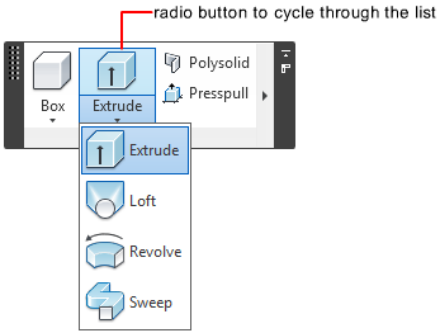
Checkboxes

Checkboxes allow you to toggle an option on or off. A third, indeterminate state occurs if the setting differes for multiple objects.



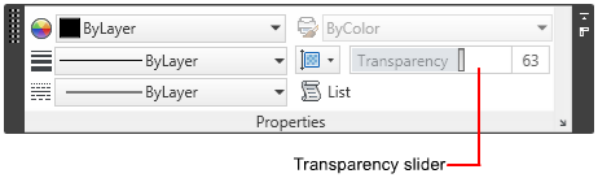
Radio Buttons

Depending on the available space in the vertical or horizontal ribbon, radio buttons can collapse into a single button. A single radio button works as a toggle, allowing you to cycle through each item in the list, or as a split button, where the top half of the radio button is a toggle button and clicking on the arrow icon in the lower half displays a drop-down of all items in the list.



Sliders

When an option can be executed with varying intensity, the slider allows you to control the setting from lower to higher, or reverse.



See also:

- Create Task-Based Workspaces on page 189

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.

To minimize the ribbon

1. The first button toggles the between the full ribbon state, the default ribbon state, and the minimize ribbon state.
2. The second drop-down button allows you to select the minimize ribbon state. These are the four minimize ribbon states:
   - **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.
   - **Minimize to Panel Buttons:** Minimizes the ribbon so that only tab titles and panel buttons are displayed.
   - **Cycle Through All:** Cycles through all four ribbon states in the order, full ribbon, minimize to panel buttons, minimize to panel titles, minimize to tabs.

To display or hide a ribbon panel

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

NOTE Some tabs include additional panels that are hidden by default.

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.

RIBBONBGLOADRIBBONCONTEXTSELECT

Controls whether ribbon tabs are loaded into memory by a background process during idle processor time.

RIBBONCONTEXTSELECT

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

RIBBONCONTEXTSELLIM

Limits the number of objects that can be changed at one time with the ribbon property controls or a contextual tab.

RIBBONDOCKEDHEIGHT

Determines whether the horizontally docked ribbon is set to the height of the current tab or a predetermined height.

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.

Customize the Ribbon

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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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

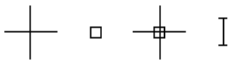
Controls the display of the Table toolbar.

Cursors in the Drawing Area

In the drawing area, the appearance of the cursor changes depending on what you are doing.

- If you are prompted to specify a point location, the cursor appears as *crosshairs*
- If you are prompted to select an object, the cursor changes to a small square called a *pickbox*
- When you are not in a command, the cursor appears as a combination of the crosshairs and pickbox cursors
- If you are prompted to enter text, the cursor appears as a vertical bar

In the following illustrations, these cursors are displayed in order.



You can change the size of the crosshairs and pickbox cursors in the Options dialog box (the OPTIONS command). For more information on customizing these and other interface elements, see Set Interface Options on page 149.

Quick Reference

Commands

OPTIONS

Customizes the program settings.

System Variables

CURSORSIZE

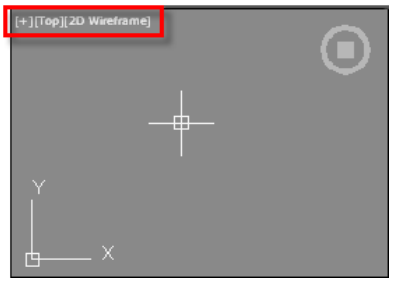
Determines the size of the crosshairs as a percentage of the screen size.

PICKBOX

Sets the object selection target height, in pixels.

Viewport Controls

Viewport controls are displayed at the top-left corner of each viewport, and provide a convenient way of changing views, visual styles, and other settings.



The labels display the current viewport settings. For example, the labels might read

[+] [Top] [2D Wireframe]

You can click within each of the three bracketed areas to change the settings.

- Click + to display options for maximizing the viewport, changing the viewport configuration, or controlling the display of navigation tools.
- Click Top to choose between several standard and custom views.
- Click 2D Wireframe to choose one of several visual styles. Most of the other visual styles are used for 3D visualization.

See also:

- Save and Restore Views on page 280
- Use a Visual Style to Display Your Model on page 294

Quick Reference

Commands

VIEW

Saves and restores named model space views, layout views, and preset views.

VSCURRENT

Sets the visual style in the current viewport.

System Variables

VPCONTROL

Controls whether the menus for viewport tools, views, and visual styles that are located in the upper-left corner of every viewport are displayed.

The ViewCube Tool

The ViewCube is a handy tool to control the orientation of 3D views.

This tool is available in most Autodesk products, and provides a common experience when you switch between products.



Alternatively, you can use the 3DORBIT command to drag 3D views, and right-click for additional 3D viewing options.

See also:

- Use 3D Navigation Tools on page 325

Quick Reference

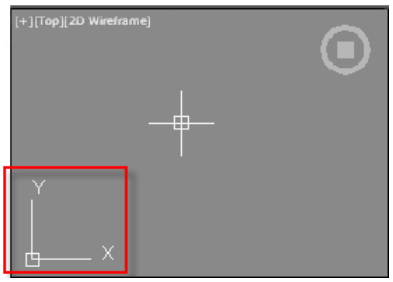
Commands

NAVVCUBE

Indicates the current viewing direction. Dragging or clicking the ViewCube tool rotates the scene.

The UCS Icon

The drawing area displays an icon representing the XY axis of a rectangular coordinate system called the User Coordinate System, or UCS.



You can select, move, and rotate the UCS icon to change the current UCS. The UCS is useful in 2D, and essential in 3D. For more information about the UCS, see Work with the User Coordinate System (UCS) on page 591.

You can control whether the UCS icon is visible, and change its appearance, with the UCSICON command, Properties option.

See also:

- Overview of the User Coordinate System (UCS) on page 591
- Control the User Coordinate System (UCS) on page 594
- Control the Display of the User Coordinate System Icon on page 603

Quick Reference

Commands

PLAN

Displays an orthographic view of the XY plane of a specified user coordinate system.

UCS

Sets the origin and orientation of the current UCS.

UCSICON

Controls the visibility, placement, appearance, and selectability of the UCS icon.

Other Tool Locations

4

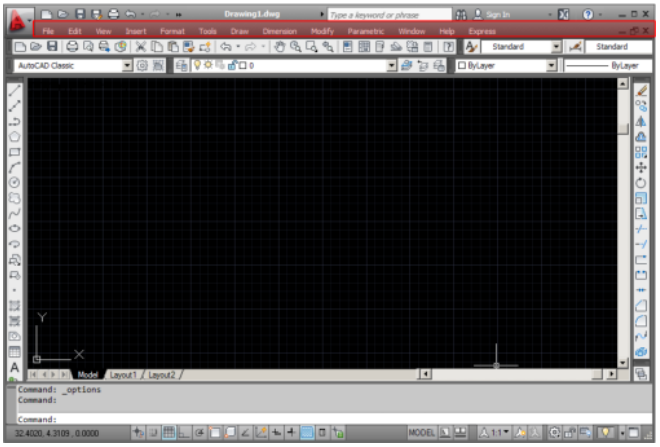
Use common tools in the classic menu bar, toolbars, tool palettes, status bars, shortcut menus, and Design Center to find more commands, settings, and modes.

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. The classic menu bar is displayed by default in the AutoCAD Classic workspace.

NOTE The classic menu bar is turned off by default in both the 2D Drafting and Annotation and 3D Modeling workspaces.



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 189
- "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.

Quick Reference

System Variables

MENUBAR

Controls the display of the menu bar.

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 22.

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.

NOTE You can choose from a list of toolbars from the View tab, Windows panel on the ribbon.

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 189
- "Toolbars" in the *Customization Guide*
- Ribbon in the *Customization Guide*

To display a toolbar

1. Click View tab ➤ Windows panel ➤ Toolbars.
2. Select a toolbar from the list.

   TIP You can also right-click any toolbar and select a toolbar from the shortcut menu.

Quick Reference

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.

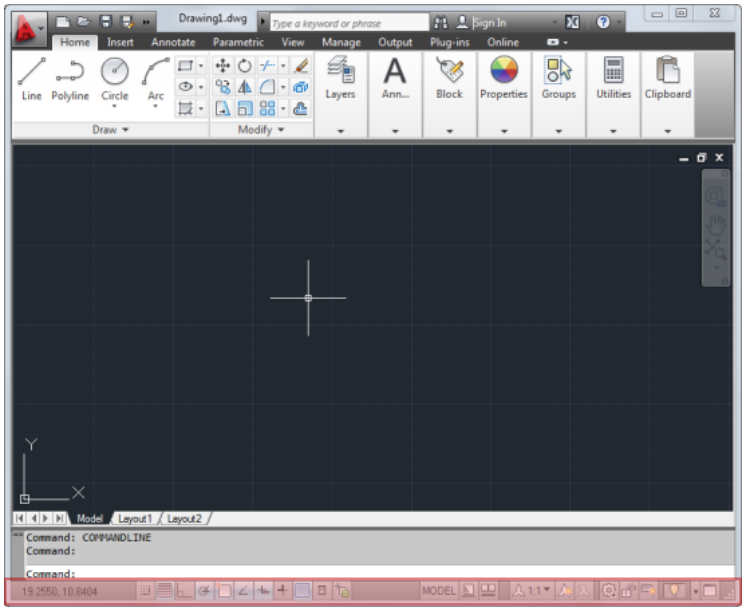
Status Bars

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, 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. You can also display tools for scaling annotations.

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

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

1. Right-click in an unused area of the application 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.

To control the display of buttons on the status bar

1. Right-click in an unused area of the application status bar.
2. In the Status Bar drop-down, select any button name to change the display.

To control the display of cursor coordinates on the status bar

1. Right-click in an unused area of the application status bar.
2. In the Status Bar drop-down, select or clear Cursor Coordinate Values.

Quick Reference

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.

WORKSPACELABEL

Controls whether to display the name of the current workspace in the status bar.

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

1. Right-click in an unused area of the application status bar.
2. In the Status Bar drop-down, click Drawing Status Bar.

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.

Quick Reference

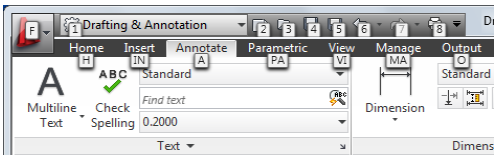
System Variables

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.

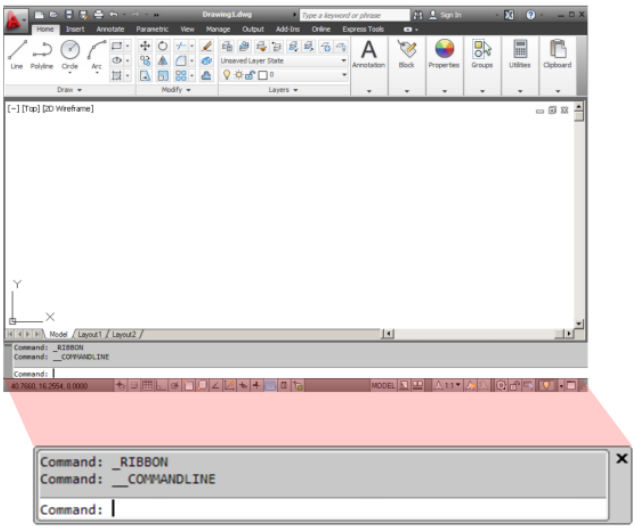
When you select a keytip, more keytips are displayed for that tool.

The Command Line 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 then press Enter or Spacebar.
- To repeat the previous command, press Enter or Spacebar without entering a command.

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

Display Valid Commands and System Variables

By default, AutoCAD automatically completes the name of a command or system variable as you type it. Additionally, a list of valid choices is displayed from which you can choose. Use the AUTOCOMPLETE command to control which automatic features that you want to use.

If the automatic completion feature turned off, you can type a letter on the command line and press TAB to cycle through all the commands and system variables that begin with that letter. Press Enter or Spacebar to start the command or system variable.

Enter Alternate Names of Commands

Some commands also have alternate names. For example, instead of entering **line** to start the LINE command, you can enter **l**. Alternate 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*.

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 location in the drawing area.

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

To use a command transparently, 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 completed. 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 within the command line.
2. Click Recent Commands.
3. Click the command that you want to use.

To change the automatic completion options for the command line

1. Right-click anywhere within the command window.
2. On the menu, click AutoComplete.
3. Click an option.

Quick Reference

AUTOCOMPLETE

Controls what types of automated keyboard features are available at the Command prompt.

MULTIPLE

Repeats the next command until canceled.

OPTIONS

Customizes the program settings.

PASTECLIP

Pastes objects from the Clipboard into the current drawing.

System Variables

AUTOCOMPLETEDELAY

Controls the amount of time that elapses before automated keyboard features display at the Command prompt.

AUTOCOMPLETEMODE

Controls what types of automated keyboard features are available at the Command prompt.

CMDNAMES

Displays the names of the active and transparent commands.

TOOLTIPSIZE

Sets the display size for drafting tooltips, and for automatic completion text at the Command prompt.

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.

Quick Reference

SETVAR

Lists or changes the values of system variables.

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 55
- Use Dynamic Input on page 619

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 View tab ➤ Windows 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.

To display the text window

- Click View tab ➤ Windows panel ➤ Text Window. 

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

Quick Reference

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 command history log file opened by LOGFILEON.

LOGFILEON

Writes the contents of the command history 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 command history are written to a log file.

LOGFILENAME

Specifies the path and name of the command history log file for the current drawing.

LOGFILEPATH

Specifies the path for the command history log files for all drawings in a session.

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

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.

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 View tab ➤ Palettes panel ➤ Command Line.
- 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 149.

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:

Quick Reference

COMMANDLINE

Displays the Command Line window.

COMMANDLINEHIDE

Hides the Command Line window.

System Variables

PALETTEOPAQUE

Controls whether palettes can be made transparent.

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.

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

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 planar to the screen.

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 user input history.

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.

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)
- Tables
- Lights
- Cameras
- Visual styles from the Visual Styles Manager
- Materials from the Materials Browser palette

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 hatch tool to override the regular scale setting when you use the tool. (An auxiliary scale multiplies your current scale setting by the plot or dimension scale.)

Blocks inserted 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 91.

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 72
- Customize Tool Palettes on page 76
- Create Task-Based Workspaces on page 189
- Add Content with DesignCenter on page 91
- Specify the Behavior of Dockable Windows on page 175

To open a block on a tool palette in the Block Editor

1. If the Tool Palettes window is not already displayed, click View tab ➤ Palettes panel ➤ Tool Palettes. 
2. On the block icon's shortcut menu, 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.

To add or remove a tool flyout

1. On a tool palette, right-click the geometric object or 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.

To customize a tool flyout

1. On a tool palette, right-click the geometric object 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 hide or show on the flyout. Click OK. (You must select at least one tool.)
4. In the Tool Properties dialog box, click OK.

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 in the Rotation box of he 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.

Quick Reference

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.

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 or the Customize User Interface (CUI) Editor to a tool palette.

NOTE You cannot drag commands from the Quick Access toolbar 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 Quick Access 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

NOTE You cannot drag commands from the Quick Access toolbar.

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, on the ribbon, click View tab ➤ Windows panel ➤ Toolbars ➤ and then a loaded customization group and the toolbar to display. You can also click Tools ➤ Toolbars from the menu bar and select a toolbar from the list.

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

   NOTE Even though you will not 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 button on the pointing device, move the cursor to the place on the tool palette where you want the tool.

   A horizontal line appears to indicate where the tool will be located.

4. Release the button on the pointing device.

5 In the Customize dialog box, click Close.

To create a command tool from the Customize User Interface (CUI) Editor

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 button on the pointing device, move the cursor to the place on the tool palette where you want the tool.

A horizontal line appears to indicate where the tool will be located.

3 Release the button on the pointing device.

4 In the CUI Editor, click Close.

To create a command tool that executes a custom command sequence (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 button on the pointing device, move the cursor to the place on the tool palette where you want the tool.

3 Release the button on the pointing device.

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

5 In the Tool Properties dialog box, change the name and description to appropriate strings based on the macro, 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, ObjectARX command, a VBA macro, or a script.

7 Click OK.

To use a command tool

1 On a tool palette, click a command tool or drag and drop the tool into the drawing that you want to use.

2 Follow any Command prompts that are shown.

Quick Reference

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.

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.

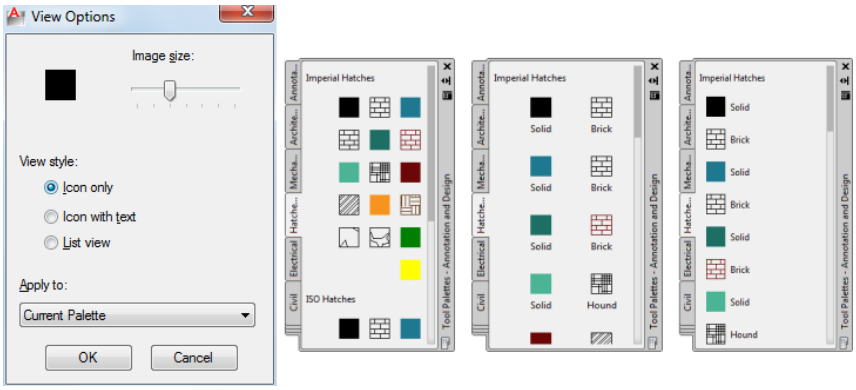
You can dock the Tool Palettes window on the right or left edge of the application window. Hold the Ctrl key if you want to prevent docking as you move the Tool Palettes window.

Tool palette settings are saved with your profile.

The Tool Palettes window shortcut menu includes

- **Allow Docking.** Toggles the ability to dock or anchor the palette window. If this option is selected, the palette window can be docked when you drag it over a docking area at the side of the application window. 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 window to an anchor tab to the left or right side of the application window. The palette window rolls open and closed as the cursor moves across it. When an anchored palette window is open, its content overlaps the drawing area. An anchored palette window cannot be set to stay open.

- **Auto-hide.** Controls the display of the palette window when it is floating. When this option is selected, only the title bar of the palette window is displayed when the cursor moves outside the palette window. When this option is cleared, the palette window stays open continuously. You can display the title bar of palette window as either icons or text from the shortcut menu of the title bar.
- **Transparency.** Sets the transparency of the palette window so it does not obscure objects under it.
- **Views.** Changes the display style and size of the icons on a tool palette.



See also:

- Specify the Behavior of Dockable Windows on page 175

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, under General, adjust the level of transparency for the Tool Palettes window when it does not have focus.

3 Under Rollover, adjust the level of transparency for the Tool Palettes window when it has focus.

4 Click OK.

NOTE Transparency is available when the Tool Palettes window is undocked or anchored.

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 Palette or All Palettes.

4 Click OK.

Quick Reference

CUI

Manages the customized user interface elements in the product.

TOOLPALETTES

Opens the Tool Palettes window.

TOOLPALETTESCLOSE

Closes the Tool Palettes window.

System Variables

PALETTEOPAQUE

Controls whether palettes can be made transparent.

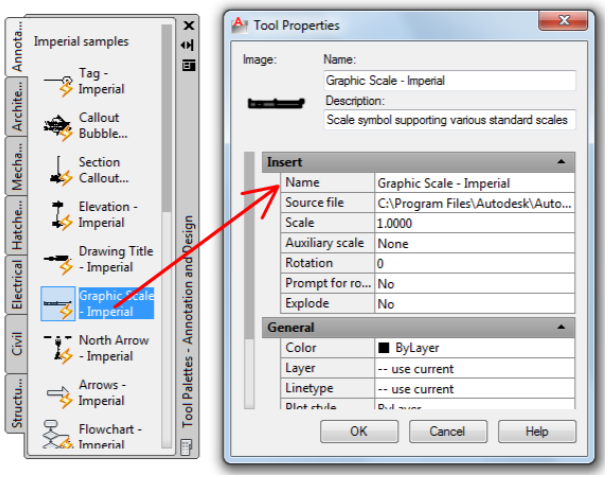
TPSTATE

Indicates whether the Tool Palettes window is open or closed.

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 click Properties to display the Tool Properties dialog box. The Tool Properties dialog box has the following categories of properties:

- **Command.** Controls the display of a tool flyout and the tools command string.
- **Camera and Clipping.** Controls the view that a camera defines.
- **Attenuation, Rendered Shadow Details, and Geometry.** Controls the lighting characteristics for a light tool.
- **Face Settings, Lighting, and Environment Settings, and Edge Settings.** Controls the visual characteristics for a visual style tool.
- **Table.** Controls the table style, numbers of rows and columns, and flow direction.
- **Insert, Attach, 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.

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 transparency. 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 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, Attach, 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.

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.

To change the image of a tool on a tool palette

1 On a tool palette, right-click a tool. Click Specify Image.

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

3 Click Open to assign 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 restore 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 Remove Specified 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.

Quick Reference

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.

Customize Tool Palettes

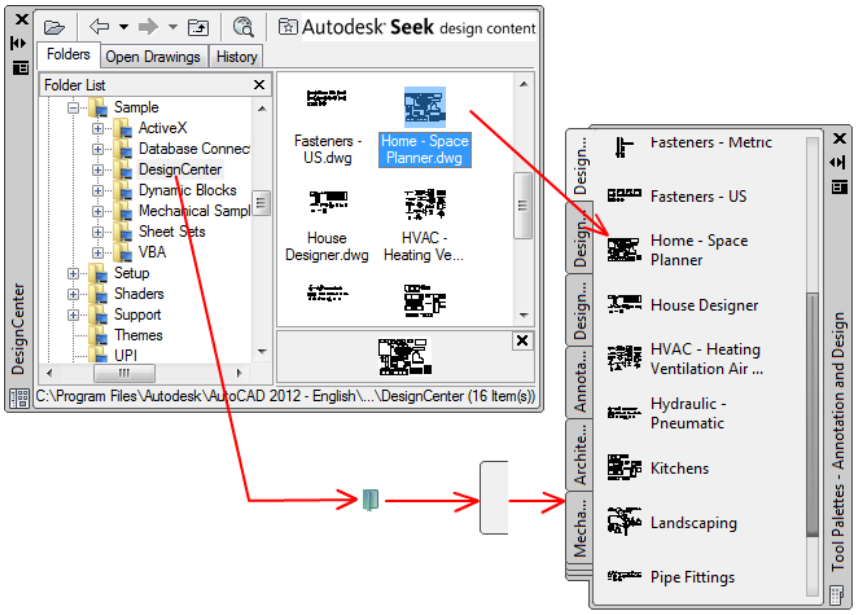
You can add tools to a tool palette with several methods.

You can create new tool palettes using the Properties button on the title bar of the Tool Palettes window and 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; tables; lights; cameras; visual styles from the Visual Styles Manager; materials from the Material Browser.
- 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 toolbar buttons to a tool palette.
- 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 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 one of the following locations:

- Windows XP: *<drive>:\Documents and Settings\<user name>\Application Data\Autodesk\AutoCAD 2012\<release>\<language>\Support\ToolPalette\Palettes*
- Windows Vista or Windows 7: *<drive>:\users\<user name>\AppData\Roaming\Autodesk\AutoCAD 2012\<release>\<language>\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 in-place text box, enter a name for the new palette.

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 tab 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 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.

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.

Quick Reference

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.

Organize Tool Palettes

Organize tool palettes into groups and control which groups are displayed.

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

When you set the Hatch Pattern group current, only the tool palettes in that group are displayed in the Tool Palettes window.

Location of Tool Palettes and Tool Palette Groups

The default path for tool palette files is set on the Files tab of the Options dialog box, under Tool Palettes File Locations. Tool palettes can be stored in multiple locations, allowing you to use both user-created and company specific tool palettes.

Tool palette groups are saved in the current AutoCAD profile.

For information on sharing tool palettes and tool palette groups, see Save and Share Tool Palettes and Tool Palette Groups on page 82.

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 in the lower, blank area. Click New Group.

   NOTE If there are no groups listed in the Palette Groups area, you can also create a group by dragging a tool palette from the Palettes area into the Palette Groups area.

3. Enter a name for the tool palette group.
4. Click Close.

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 Palettes area into a group in the Palette Groups area.
3. Click Close.

To copy 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.
   A copy of the tool palette is displayed in the new location.
4. Click Close.

To display a tool palette group

1. Right-click the title bar of the Tool Palettes window.
2. Click the name of the tool palette group that you want to display.

To display all tool palettes

- Right-click the title bar of the Tool Palettes window. Click All Palettes.

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 a tool palette group. Click Delete.
3. Click Close.

Quick Reference

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.

Save and Share Tool Palettes and Tool Palette Groups

Export and import tool palette files (XTP) and tool palette group files (XPG) to share with others.

NOTE For read-only tool palette files, 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.

Export Tool Palettes and Tool Palette Groups

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 in the exported tool palette. The folder is created when you export a tool palette that contains any of the following:

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

WARNING Avoid copying tool palette (ATC) files between releases. Doing so can cause potential problems when migrating to a future release. For more information, see Migrate Custom Settings in the *Customization Guide*.

Import Tool Palettes and Tool Palette Groups

If you are importing tool palettes and tool palette groups, import all tool palettes first. Once all tool palettes are imported, you can them import the exported tool palette groups.

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

Tool palettes can be used only in the AutoCAD release in which they were created. For example, you cannot use a tool palette that was created in AutoCAD 2012 with AutoCAD 2005. You can migrate tool palettes from previous releases to the current release using Migrate Custom Settings.

WARNING If you share tool palettes with someone who is not using the same AutoCAD-based product or release in which they were created, it is possible that the tools might not work properly or be accessible.

See also:

- Save and Restore Interface Settings (Profiles) on page 193

To export a tool palette or tool palette group

1. Click Manage tab ➤ Customization panel ➤ Tool Palettes. 
2. In the Customize dialog box, do one of the following:
   - **Export a tool palette.** Under Palettes, right-click a tool palette. Click Export.
   - **Export a tool palette group.** Under Palette Groups, right-click a tool palette group. Click Export or Export All to export all tool palette groups.
3. In the Export <*Item*> dialog box, specify a file name and location. Click Save.
4. Click Close.

To import a tool palette or tool palette group

1. Click Manage tab ➤ Customization panel ➤ Tool Palettes. 
2. In the Customize dialog box, do one of the following:
   - **Import a tool palette.** Under Palettes, right-click and click Import.
   - **Import a tool palette group.** Under Palette Groups, right-click any tool palette group or an empty area, and click Import.
3. In the Import <*Item*> dialog box, locate the file you want to import. Click Open.
4. Click Close.

Quick Reference

CUSTOMIZE

Customizes tool palettes and tool palette groups.

TOOLPALETTES

Opens the Tool Palettes window.

TOOLPALETTESCLOSE

Closes the Tool Palettes window.

DesignCenter

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

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.

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.

See also:

- Specify the Behavior of Dockable Windows on page 175

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.

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.

To display and hide the DesignCenter tree view

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

Quick Reference

ADCENTER

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

ADCNAVIGATE

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

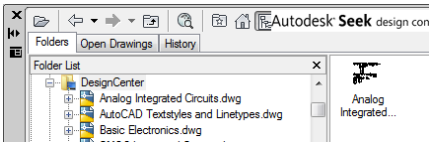
System Variables

ADCSTATE

Indicates whether the DesignCenter window is open or closed.

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.

DesignCenter Tabs and Autodesk Seek

The Open Drawings, History, and DC Online tabs, and Autodesk Seek provide alternate methods of accessing 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.
- **Autodesk Seek.** Provides content from the Autodesk Seek web page including blocks, symbol libraries, manufacturer's content, and online catalogs. Autodesk Seek is not available in all regions and products.

NOTE The DesignCenter Online (DC Online tab) is turned off by default. You can enable it from the CAD Manager Control utility on page 97. However, it is not available in English-language versions of the product.

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.
   - **Autodesk Seek.** Provides content from the Autodesk Seek web page including blocks, symbol libraries, manufacturer's content, and online catalogs. Autodesk Seek is not available in all regions and products.

NOTE The DesignCenter Online (DC Online tab) is turned off by default. You can enable it from the CAD Manager Control utility on page 97. However, it is not available in English-language versions of the product.

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.

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.

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.

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.

Quick Reference

ADCENTER

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

ADCNAVIGATE

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

SEEK

Opens a web browser and displays the Autodesk Seek home page.

System Variables

ADCSTATE

Indicates whether the DesignCenter window is open or closed.

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.

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.

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.

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.

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.)

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.

To open a block from the DesignCenter window in the Block Editor

1. Click View tab ➤ Palettes panel ➤ DesignCenter. 
2. On the block icon's shortcut menu, click Block Editor.

Quick Reference

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.

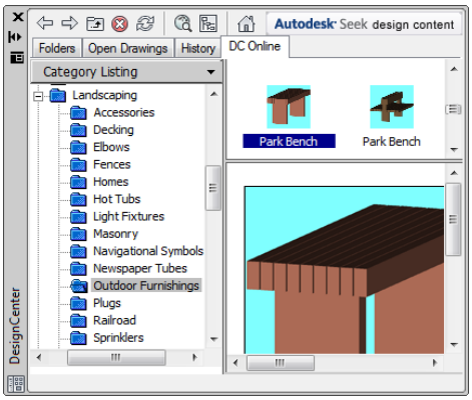
Retrieve Content from the Web with DesignCenter Online

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

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.

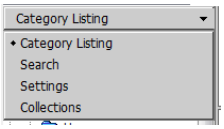


NOTE The DesignCenter Online (DC Online tab) is disabled by default. You can enable it from the CAD Manager Control utility on page 97. However, it is not available in English-language versions of the product.

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.

Quick Reference

ADCENTER

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

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 97. However, it is not available in English-language versions of the product.

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.

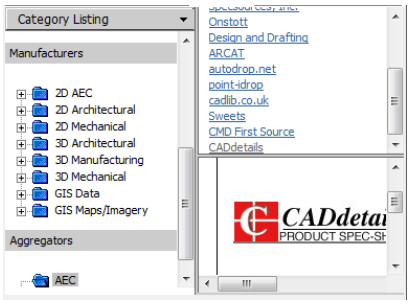
The category folders are displayed in the left pane of the window.

Retrieve Content from the Web

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

Browse for Content

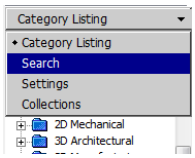
When you use the Category Listing view, you can click the folders in the left pane to view their contents. These folders may contain other folders.



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 on page 2122

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

ADCENTER

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

Content Explorer

With Content Explorer, you can search for and access design files and design objects locally or on network servers without leaving the CAD environment.

Overview of Content Explorer

Content Explorer aggregates design content from different folders that have been specified for monitoring by the Content Service.

With Content Explorer you can index design content for quick access, catalog the objects in each file, and search for content in local folders, network folders, and the Autodesk Seek Library.

With Content Explorer, you can

- Browse and search design content in local folders, network folders, and the Autodesk Seek Library and immediately access files and objects from within the AutoCAD environment
- Browse into DWG files and access and insert blocks, layers, linetypes, styles, etc.
- Search for objects, all text, attributes (including block attributes), and files in specified local and network folders.

NOTE Specify the Autodesk Seek Library download folder as a watched folder and any downloaded files will also be indexed.

- Pinpoint specific block references or text strings and automatically navigate to the containing file
- Open any file from Content Explorer and zoom to any attribute within it
- Save search parameters for instant access to design content that meets your design needs
- Customize how search results are displayed to streamline your work environment

NOTE Folders on the local computer or a network computer must be specified as watched folders before their contents can be cataloged and indexed.

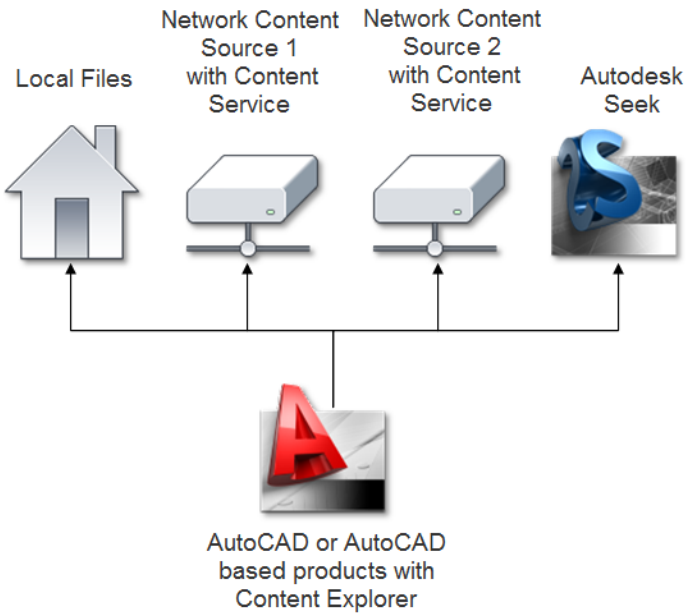
Understand the Relationship between Content Explorer and Content Service

Content Service is a companion component to the Content Explorer search client installed with AutoCAD and the AutoCAD verticals.

Content Service is an indexing service that monitors the activity in watched folders on the computer where it is installed. While monitoring these watched folders, Content Service continuously updates an index with critical metadata associated with all design files and design objects in those watched folders.

When you perform a search with Content Explorer, the index maintained by Content Service is examined for matches to the search criteria. Since the index is continually updated by Content Service, the search results appear immediately. With this real-time search capability, you can create custom searches for thousands of design files and design objects, and instantly display only those files and objects you need for your design.

If you plan to search folders on several different network computers, Content Service must be installed on each network computer and the folders added to the watched folder list for indexing. Any machine running AutoCAD or an AutoCAD vertical with Content Explorer can access the network machines running Content Service for real-time search results.



NOTE Refer to the Content Service help for more information.

Personal vs. Public Content Service

Content Service is installed as either a personal service or a public service, depending on how it will be used.

Personal Service

- Whenever AutoCAD is installed, Content Service is automatically installed on the local system as a personal service. This means that you are going to be using AutoCAD to access design files and will be managing watched folders with Content Explorer in AutoCAD.

Public Service

- If you do not intend to use AutoCAD on a network machine, but you want to add watched folders on that machine, then you will use Content Service as a public service. This means that you must manually install Content Service on the network machine. Watched folders can then be managed with the Autodesk Content Service - Administration Console. However,

AutoCAD users on other machines will be able to search them and access them.

NOTE Refer to the Content Service help file for more information on using Content Service on a network machine.

Understand the Content Explorer Window

Content Explorer is a powerful indexing and searching application with a compact and efficient design similar to the AutoCAD External References Manager window.

Just as with other secondary windows in AutoCAD, you can configure transparency, auto-collapse, and docking preferences. See Specify the Behavior of Dockable Windows on page 175 for more information on configuring the Content Explorer window.

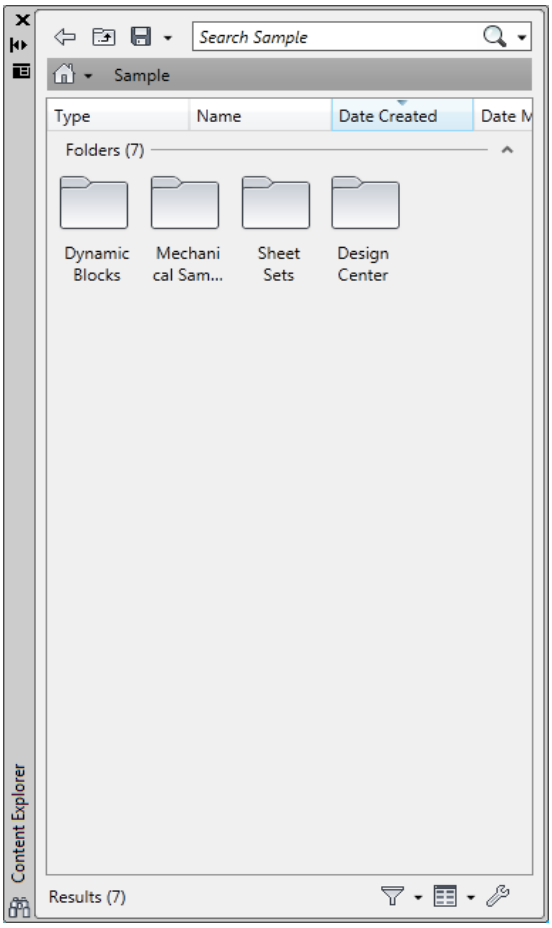


Image	Feature	Description
	Back	Recalls the last results you viewed. The last results can be from browsing, searching, or recalling a saved search. For example, you've performed a search that returned multiple results. You double-clicked on a file to browse its contents. Click Back to return to the search results.

Image	Feature	Description
		You can click Back until you reach the first search performed since starting the Content Explorer session.
	Up	Climbs the breadcrumbs path one step at a time. You can click Up until you reach the top level for the specified content source.
	Home	Click Home to return to the top level of the Home. This view shows the local folders that are currently watched and indexed. Watched folders on the local machine can be added and removed at the top level of Home.
	Autodesk Seek	Click Autodesk Seek to switch to the Autodesk Seek library. This content source gives you access to the products available on the Autodesk Seek web site.
	Network Machine	Click Network Machine to return to the top level of a Network Machine. This view shows the network folders that are currently watched and indexed. Watched folders on the network machine can be added and removed on the Content Service Administration Console. See the Content Service help for more information.
	Search field	Enter a search string in the Search field to locate the files and objects that meet the specified criteria.

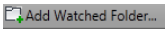
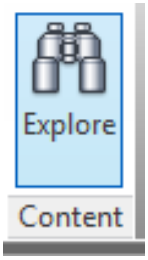
Image	Feature	Description
		Only the current content source is searched. Results are displayed based on the current filter settings.
	Recent Searches	Click Search to perform a search. Click the Search drop-down arrow to display previous searches.
	Saved Searches	Click Saved Searches to save the current search string. Select the saved search drop-down to display the list of saved searches. You can also delete saved searches, edit the names of existing searches, and save new searches in this context menu.
	Add Watched Folder	Click Add Watched Folder to add a new local folder to Home for monitoring and indexing. You may also add a folder by dragging and dropping a folder from your Windows Explorer window into Home.
	sults and Pending display	Folders are constantly indexed for any changes, additions, or deletions. When a folder is in the process of being indexed and not all of the information is available yet, a pending icon will appear in this window.
	Filter **NOTE** The Filter feature is disabled when browsing and searching in Autodesk Seek.	Click Filter to enable your current filter settings. Click the filter drop-down to specify which types of files and objects are displayed when you browse or search a folder. When a filter is active, the icon highlights to let you know that not

Image	Feature	Description
		everything is being displayed, based on your filter requisites.
	View Options	Click View Options to cycle through icon size. Click the View Options drop-down arrow to configure the icon size, whether the folders are displayed in thumbnail or detailed view, which labels are displayed, and how the results are grouped.
	Configure Settings	Click Configure Settings to add, remove, enable, and disable content sources.

To Launch Content Explorer

Access Content Explorer with one of these methods:

- Go to the Plug-ins tab and click Explore.



- Enter CONTENTEXPLORER on the command line.

To Close Content Explorer

Close Content Explorer with one of these methods:

- Select the Close icon in the Content Explorer window.
- Enter CONTENTEXPLORERCLOSE on the command line.

You can also toggle the window so that it is not displayed by clicking the Explore button again.

To Determine State of Content Explorer Window

- Enter CONTENTEXPLORERSTATE on the command line to retrieve a read-only variable indicating whether the Content Explorer window is open or closed.

Understand How Content Is Indexed

When a folder is specified as a watched folder, the Content Service, a companion component to Content Explorer, indexes all of the design files and design objects in the folder, including associated metadata.

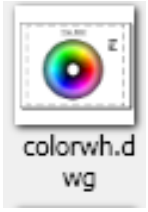
Content Service continues to monitor the folders in the background and updates the index to reflect any changes to design objects and files, including new objects and folders, edits to existing files, and deletions.

When Content Explorer performs a search, it references the index maintained. Since the index is constantly updated with changes, Content Explorer obtains real-time results immediately.

Indexing Stages

The indexing occurs in three stages, with deeper indexing occurring with each stage. The stages can be determined by the thumbnails displayed in the Content Explorer window.

Stage	Icon Displayed	Description
1		File has been discovered by the indexer. By the end of this stage, file name and standard Windows properties have been extracted. You can search for file names and Windows properties, but you cannot explore the content within the file. **NOTE** Since all text and attributes in the files are indexed, the initial index of the files contained in the watched folders may take some time to complete.

Stage	Icon Displayed	Description
2		Thumbnail and DWG-specific file properties have been discovered. During this stage, all objects and any text in the drawing, including block attributes, are being indexed. You can now search for DWG properties, but you cannot explore the content within the file.
3		File is completely indexed. You can now explore or search for content within the file.
!		There was an issue while indexing this file. Select the file and press CTRL+ I for more information regarding the issue.

What Is Indexed

All text and attributes in the files are indexed, including block attributes. In addition, the following objects are indexed when a folder is selected for monitoring:

- Block definitions
- Block references
- Dimstyles
- Layers
- Layouts
- Linetypes
- Multileaderstyles

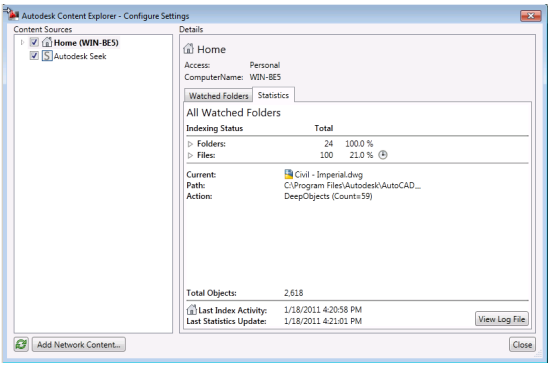
- Tablestyles
- Textstyles
- Xrefs
- Hyperlinks

View Indexing Statistics in Content Explorer

You can review the indexing status of your files on the Statistics tab of the Configure Settings dialog.

To review the indexing statistics in Content Explorer

1. In Content Explorer, click Configure Settings.
2. On the Configure Settings dialog, select either your local computer or a network content source and click the Statistics tab.



3. Review the percentage of folders that have been indexed and the percentage of files that have been indexed in those folders.

   For example, in the image above, 100 percent of the folders have been indexed without error but only 21 percent of the files have been indexed so far.

4. Click the View Log File button to review a more detailed list of indexing events and determine any issues.
5. If the indexing service stalled without any current actions, remove the watched folder and re-add it for indexing.

   This action will start the indexing process over and will resolve most issues.

NOTE The Statistics tab is also available in the Autodesk Content Service Administration Console. Refer to the Autodesk Content Service Administration Console help for more information.

Understand Watched Folders

A watched folder is a folder which has been selected for monitoring and indexing by the Content Service.

A content source may be the local machine, Autodesk Seek, or designated network location. When you search a content source that is a local or network machine, all watched folders are searched. Any design files or objects in those folders that match the search criteria are returned.

Watched folders can exist on the local machine where AutoCAD or an AutoCAD vertical is installed, or on a network computer running Content Service. You can configure which folders are watched in two different places, depending one whether the folders are local or on a network.

Local Folders

If you are adding local folders to the search index for watching or if you will be monitoring Autodesk Seek download folders, then you can configure the watched folders using Content Explorer.

Network Folders

Folders on a network computer must be indexed and monitored by Content Service. Content Service runs in the background, monitoring activities in the watched folders and updating a single index to reflect any changes. You can add and delete watched folders on a network machine in the Autodesk Content Service-- Administration Console.

There are several rules regarding how Content Service watches folders on a network computer. It is important to understand these rules before selecting a folder as a watched folder.

Rules for Watched Folders on a Network Computer

- The target folder must be configured as shared. If the folder is not shared, it will not show up on the Select a Folder to Watch list for selection.
- By allowing Content Service to index a shared folder, you are allowing anyone to search the contents of that watched folder, even if some of the

files are read-only. However, users will not be able to access read-only content; they will only see content if they browse to it or if it meets search criteria.

- If the top level of a folder hierarchy is shared and specified as a watched folder, all of the subfolders are watched and indexed, regardless of the share setting. If you do not want content inside a folder hierarchy to be shared, move it to a private location or build a new hierarchy of data to share.
- If a top level folder is shared, the contents inside are indexed but parallel data to that top level folder is not indexed.

Add Watched Folders

You can add folders to the watched folders list in Content Explorer or via the Autodesk Content Service Administration Console. Generally, you should add folders to the watched folder list through Content Explorer. However, if you need to add folders to a watched list and AutoCAD or an AutoCAD vertical are not installed, then you can use the Autodesk Content Service Administration Console to specify watched folders.

NOTE Refer to the Autodesk Content Service help for more information.

You can add a folder for watching through Content Explorer based on the content source that has been selected. Refer to Manage Content Sources on page 113 for more information.

See also:

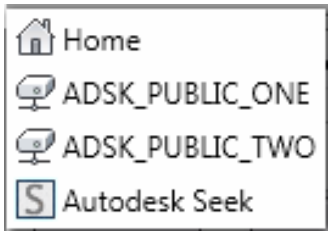
- Understand How Content Is Indexed on page 109
- Manage Content Sources on page 113
- Understand the Home Content Source on page 117
- Understand the Network Content Source on page 122

Manage Content Sources

Content sources are locations where design content is stored.

A content source can be your local drive, a network computer, or a web site like Autodesk Seek.

You can search only one content source at a time. When searching your local drive or a network computer, you can search all watched folders for that content source at the same time or drill down to a specific folder for a smaller number of files and objects to search.

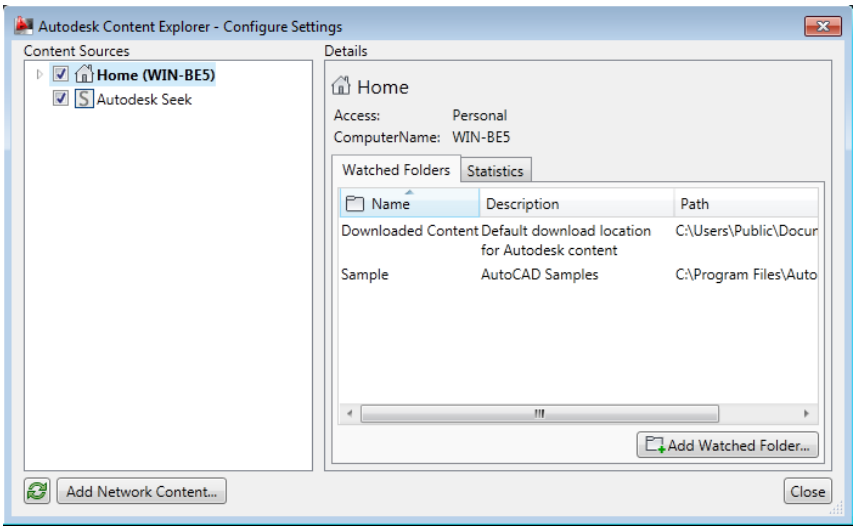


Select content sources from the source drop-down menu.

The content source type currently selected is represented by a unique icon.

Icon	Content Source
	Home. The Home content source represents all watched folders on the local drive. **NOTE** If your local machine also serves as a network machine for others, then you will only see the Network icon.
	Network. The Network content source represents watched folders on a network machine. The network machine is identified by its network name.
	Autodesk Seek. The Autodesk Seek content can be browsed, examined, and downloaded from this view.

Content sources can be added, removed, enabled, or disabled in the Configure Settings dialog box.



See also:

- Understand the Home Content Source on page 117
- Understand the Seek Content Source on page 119
- Understand the Network Content Source on page 122

To add a content source

Add a new content source to the list of available sources for searching.

1. In the Content Explorer window, click the Configure Settings icon.
2. On the Configure Settings dialog, click the Add Network Content button.
3. Enter the name of the computer on the network that you want to add to the Content Source list. This computer must be running Autodesk Content Service.

   NOTE See the Autodesk Content Service help for more information on configuring watched folders on a network computer.

4. Click OK.

The network computer is added as an available content source.

To remove a content source

Remove content sources which you no longer want to browse or search, or which are no longer available.

1. In the Content Explorer window, click the Configure Settings  icon.
2. In the Content Sources list, right-click on the content source and select Remove.

   The content source is removed from the list and is no longer available for browsing or searching.

To disable a content source temporarily

If a network machine becomes unavailable and you do not want it listed for searching, you can temporarily disable the content source.

1. In the Content Explorer window, click the Configure Settings  icon.
2. In the Content Sources list, clear the check box next to the content source that you want to disable.

   The content source is disabled for searching and browsing, but remains on the content sources list as an available content source.

To enable a content source

1. In the Content Explorer window, click the Configure Settings  icon.
2. In the Content Sources list, select the check box next to the content source that you want to enable.

   The content source is enabled for searching and browsing.

Understand the Home Content Source

Home represents the local file system folders that are currently being indexed and monitored.

Folders added to Home are watched for file system events, such as Add, Save, and Delete. An indexing process keeps a current catalog of the files and objects located in the watched folders. You can easily locate files in the specified folders based on search criteria, such as the date last updated, the author, a property value, or the file type.

You can configure a single location as the source folder for monitoring all content. You can also create unique file locations as needed, such as an archived content folder and a current project content folder.

NOTE Network locations are not considered local data sources.

Home consists of local folders which have been identified for indexing. You can add more folders to Home for indexing and browsing, or remove those you no longer need

Configure Home by adding and removing folders. Folders in Home will be monitored for updates and indexed. When you perform a search, only the content contained in the watched folders is searched.

Feature	Description
Home content source	The Home content source lists the folders being indexed and additional information depending on the view selected.
Add Watched Folder	Click Add Watched Folder to add a new folder to the Home content source for monitoring and indexing. You may also add a folder by dragging and dropping a folder from your Windows Explorer window into the Home content source. **NOTE** Adding a folder to the Home does not move the folder or its contents. When the Add Watched Folder button is clicked, a Browse for Folder dialog box opens for easy navigation to the desired folder. You can also create a new folder in the hierarchy by clicking the New Folder

Feature	Description
	button on the Browse for Folder dialog box. **NOTE** Folders already added to Home for indexing are displayed with a red check mark and are grayed out.
Remove	Right-click on a folder and select Remove to remove the selected folder location from the Home content source. You can also drag a folder from the Home content source and drop it outside of the window to remove it from the Home. A folder which has been removed is no longer indexed. **NOTE** Removing a folder from Home does not delete the folder from the file system.

See also:

- Understand Watched Folders on page 112
- Manage Content Sources on page 113
- Understand the Seek Content Source on page 119
- Understand the Network Content Source on page 122
- Basic Searching with Content Explorer on page 125

To add a folder to Home

1. Launch the Content Explorer window.
2. Click the Home  icon to go to the Home content source.
3. Click Add Watched Folder to open the Browse for Folder dialog box.
4. Navigate to the folder which you want to add to the index and click OK.

   The folder is added to Home.

   You can also click New Folder and create a new folder in the specified file hierarchy. Select the new folder and click OK to add it to Home.

NOTE Adding a folder to Home does not move the folder or its contents.

To remove a folder from Home

1. Launch Content Explorer.
2. Click the Home  icon to go to the Home content source.
3. Right-click on the folder you want to remove from Home and select Remove from the context menu.

   The folder is removed from Home and is no longer indexed.

NOTE Folders removed from Home are not deleted from the file system.

Understand the Seek Content Source

Browse and search for products on Autodesk Seek using the Seek content source in Content Explorer.

You can explore the detailed specifications for each product or select design files to download and drag-and-drop into the open drawing.

Examine Products with Autodesk Seek

Double-click on products to view more information about that product, such as the description or technical specifications. You can also determine which files are available for download.

NOTE Only .dwg files can be downloaded at this time.

Search with Autodesk Seek

When searching with the Seek content source, you can locate files by manufacturer or by product specifications. You can also insert files directly into your drawing without accessing the external web site.

Explore Autodesk Seek Online

The Seek content source displays online content found at Autodesk Seek (http://seek.autodesk.com). With the Seek content source, you can download

content and insert it directly into your drawing. You can also access the external web site by clicking View products at Autodesk Seek.

Access the products online by clicking View Products at Autodesk Seek. A web browser opens directly to the Autodesk seek site online. The results are displayed according the search you entered in Content Explorer.

NOTE Refer to the section on Autodesk Seek for more information.

Feature	Description
Seek content source	The Seek content source lets you search for online content on Autodesk Seek and insert content directly into the open drawing without ever leaving your design environment.
View Products at Autodesk Seek	Opens a separate browser that takes you directly to the Autodesk Seek site where you can explore available content.

See also:

- Use Autodesk Seek to Add and Share Drawings on page 2122
- Manage Content Sources on page 113
- Understand the Home Content Source on page 117
- Understand the Network Content Source on page 122

To explore content

1. Launch the Content Explorer window.
2. Click the Seek  icon to go to the Seek content source.
3. Enter your search parameters in the Search field.
4. Navigate to the product that you want to explore.
5. Right-click on the product and select Explore.

   The description and specifications for the product are displayed. You can also see the DWG files that are available for download.

NOTE Click the Back button to return to the previous view.

To insert and open content

1. Launch the Content Explorer window.
2. Click the Seek  icon to go to the Seek content source.
3. Enter your search parameters in the Search field.
4. Navigate to the product that you want to insert into your drawing.
5. Perform one of the following actions:
   - *To open and insert*–Right-click on the file and select Open to download and open the file.
   - **To insert** –Drag and drop the file into the CAD environment or right-click on the file and select insert.
   - **To download**–Right-click on the file and select download.

   With all of the above actions, the file is added to the download folder specified for Autodesk Seek content.

NOTE If the download folder for Autodesk Seek content is a watched folder, searches from the Home content source will include downloaded Seek content.

To access Autodesk Seek online

1. Launch Content Explorer.
2. Click the Seek  icon to go to the Seek content source.
3. Click View products at Autodesk Seek.

   A separate window launches, taking you directly to the Autodesk Seek web site.

NOTE See the section on Autodesk Seek for more information.

Understand the Network Content Source

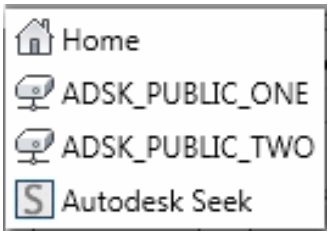
A network content source represents the file system folders that are currently being indexed and monitored on a network machine.

Folders shown in a network content source are watched for file system events, such as Add, Save, and Delete. An indexing process keeps a current catalog of the files and objects located in the watched folders. You can easily locate files in the specified folders based on search criteria, such as the date last updated, the author, a property value, or the file type.

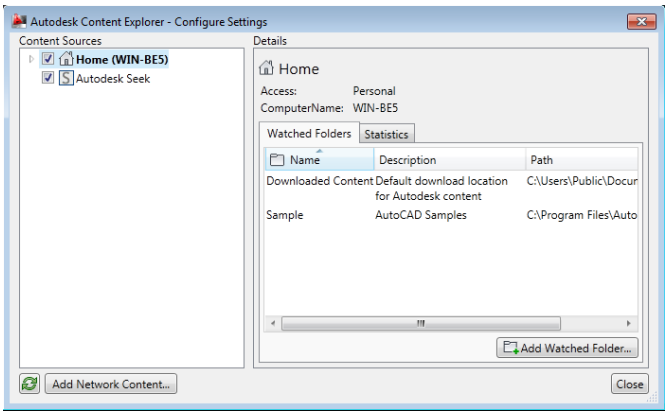
You can configure which folders are watched on the network machine in two ways:

- If AutoCAD or an AutoCAD vertical is installed on the network machine, launch Content Explorer and add the folders through the Autodesk Content Explorer- Configure Settings dialog box.
- If AutoCAD or an AutoCAD vertical is NOT installed on the network machine, launch the Autodesk Content Service Administration Console and add the folders through the Add Watched Folder dialog box.

When you search a network content source, only watched folders on that network machine are searched. If you need to search a different network machine, you must select that content source from the Content Source drop-down.



If AutoCAD or an AutoCAD vertical is installed on the network machine where you want to add watched folders, you can add and remove watched folders with Content Explorer on the Autodesk Content Explorer Configure Settings dialog box.



NOTE Folders on a network computer must be set as shared to be added as watched folders on that network computer. Only a network computer running Content Service can be added as a network content source in Content Explorer.

If the network machine does not have AutoCAD or an AutoCAD vertical installed, you can specify watched folders for that machine through the Autodesk Content Service Administration Console.

These folders must be set to shared before you can add them as a watched folder. If a folder is not configured to be shared, it will not show up on the list of available folders to add for watching.

NOTE It is recommended that you use Content Explorer to add watched folders unless you need to add watched folders on a computer where AutoCAD or an AutoCAD vertical is not installed.

See also:

- Understand Watched Folders on page 112
- Manage Content Sources on page 113
- Understand the Home Content Source on page 117
- Understand the Seek Content Source on page 119

To add or remove watched folders with Content Explorer

1. In the Content Explorer window, click the Configure Settings  icon.
2. Click Add Watched Folder to launch the Add Watched Folder dialog box.
3. Navigate to the folder that you want to add, select it, and click OK.

   The folder is added to the watched folder list.
4. To remove a folder, right-click on the folder in the Watched Folders list and select remove.

   The folder is no longer available for searching or browsing with Content Explorer.

To add or remove watched folders with the Autodesk Content Service Administration Console

1. Open Windows Explorer and navigate to the folder that you want to add as a watched folder.
2. Right click on the folder and select Properties.
3. Select the Sharing tab and verify that the folder is set to Shared. Click okay to close the Properties dialog box.
4. Launch the Autodesk Content Service Administration Console by performing one of the following steps:

   For Windows XP:

   - Go to Start Menu ➤ Programs ➤ Autodesk ➤ Content Service ➤ Content Service - Administration Console.

   For Windows 7:

   - Go to Start Menu ➤ All Programs ➤ Autodesk ➤ Content Service ➤ Content Service - Administration Console.
5. Click Add Watched Folder.
6. On the Select Watched Folders dialog box, select the folder that you want to have watched or click New to create a new folder.

NOTE If the folder you are looking for is not listed, verify that it is configured as a shared folder.

7. Click OK to add the selected folder to the watched list. Content Service will now index and monitor all activities in that folder.
8. Repeat steps 5 through 7 for each folder that you want Content Service to watch.

   New watched folders automatically appear in the specified Network Content source in Content Explorer.
9. To remove a folder from the watched list, right-click the folder and select remove.
10. Click Yes on the Remove Folder dialog box.

    The folder is no longer watched and cannot be searched or browsed with Content Explorer.

Basic Searching with Content Explorer

The search feature allows you to quickly locate files that meet your search criteria.

When you enter a search string into the search field, Content Explorer examines the index and returns the files with a file name, author, keyword, comment, name, subject, or title that meets the search criteria. Any objects with a file name that meets the search string are also returned.

In addition to file properties and objects, you can also search for text contained in:

- Multiline text
- Tables
- Fields
- Multileaders
- Dimensions
- Hyperlinks

Any block attributes which meet the search criteria are also returned.

NOTE The search feature is relative. If you have browsed to a subfolder, Content Explorer will search files and folders within the current folder only.

Search results for local and network content sources are displayed 100 to a page.

Search results for the Seek content source are displayed 20 to a page.

Search with Wild Cards

You can use wild cards when specifying search criteria.

Wild Cards

*	Represents any number of characters within a string.
?	Represents a single character within a string.

The search engine assumes a trailing asterisk (*), so you do not need to put an asterisk at the end of a search phrase.

To perform a basic search with Content Explorer

1. Enter the text string for which you want to search in the Search field.
   You can use wild cards when searching.
2. Press Enter or click Search.
   Files meeting the search criteria are displayed.

NOTE If your search results are not what you expected, try changing your filter settings or refining your search.

Advanced Searching with Content Explorer

Advanced searches can be performed using property:value pairings or boolean operators.

You can refine your searches by using string combinations and value pairings.

Use Property Name and Value Pairs

A property:value pair is the property name and specific value for which you want to search. To search for a property with a specific value, enter the data as *`property:value`* in the Search field.

For example, enter *author:jsmith* in the Search field to find all of the DWG files where the author was JSmith.

More than one property:value pair can be used in a search string. For example, you can search for all of the drawing files identified as seating which were created by jsmith by entering *objecttype:seating author:jsmith* in the Search field.

You can create custom properties and values for your files on the AutoCAD properties dialog box. See Streamline Searches with User-Created Properties in Content Explorer on page 139 for more information.

Use Attributes and Value Pairs

An attribute:value pair is the block attribute and specific value for which you want to search. To search for a block attribute with a specific value, enter the data as *attribute:value* in the Search field.

For example, enter *Designedby:JohnDoe* in the Search field to find all of the objects designed by John Doe.

Search Text for a Specific Value

With the basic search capability, you can enter a text string and the search engine will return all files with a file name, keyword, title, author, or other properties that meet the string criteria, as well as any files containing the string, and objects with names that match the text string. Sometimes this type of string search will return more results than you need.

Use the text:string pair to search only for text entities, such as leaders, fields, hyperlinks, MText, tables, or any other text that may show up on a drawing. This type of search parameter will not search file-level properties, such as Author or Title.

To search for a specific text string, enter the data as *text:string* in the Search field.

For example, enter *text:reviewed* in the Search field to find all of the files with text entities containing the string "reviewed."

Search with Boolean Operators

You can use operators to further refine your search results.

For example, to search for all of the files identified as seating that were not authored by JSmith, enter *objecttype:seating NOT author:jsmith* in the Search field.

Operators

AND	Searches for x AND y in any order. The search granny AND smith returns anything containing both words.
OR	Searches for either x OR y. The search granny OR smith returns anything containing either word.
NOT	Searches for x but NOT y. The search granny NOT smith returns anything containing granny but not smith.
" "	Searches for the exact phrase contained within the quotation marks. The search "granny smith" returns everything containing the exact phrase granny smith.

To perform and advanced search in Content Explorer

1. Enter the text string or property:value pairing for which you want to search in the Search field.

   Include any operators to further refine your search.

2. Press Enter or click Search.

NOTE If your search results are not what you expected, try changing your filter settings or refining your search.

Manage Saved Searches in Content Explorer

The saved searches feature allows you to capture a certain search string in a single click.

You can eliminate all of the steps required to search for specific files or objects by saving the search string.

For example, you can save a search that locates all of the furniture created by a specific designer. Any time you want to quickly access furniture, select the saved search. Since the index is continuously updated, any new files that meet the search criteria are displayed every time the search is recalled.

You can also delete saved searches, edit the names of existing saved searches, and save new searches on the Saved Searches drop-down menu.

Save, Edit, and Delete Searches in Content Explorer

Saving and reusing previous searches can save you time. You can manage your saved searches in the Saved Searches drop-down list.

The five most recently entered search strings are stored in the search field drop-down list. When you enter text that matches the beginning characters of a previously entered search string, that search string is recommended.

When the results of a saved search are displayed in the main view, these results are filtered and grouped based on the current settings. The sort and group features can be used to further refine how search results are displayed.

Saved searches can be deleted or the search names modified in the saved and recent searches  drop-down.

To save a search in Content Explorer

1. Perform a search and then click Save Searches  .
2. Enter a name for the saved search and press Enter.
   The search is saved.

To rename a search in Content Explorer

1. From the saved search drop-down list, click Edit  next to the recent search string that you want to rename.
2. Enter a new name for the saved search and press Enter.
   The search is renamed.

To delete a saved search in Content Explorer

- Click Delete  next to the search you want to remove from the saved searches drop-down list.

To create a saved search in Content Explorer

1. Perform a search or browse into a file.

2 Click the Save Searches  icon.

3 Enter a name for the saved search in the field. The search is added to the Saved Searches list.

To use a saved search in Content Explorer

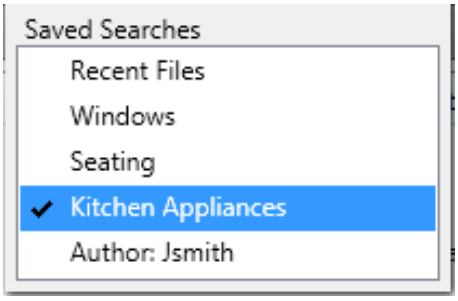
Use saved searches at any time for fast access to your indexed files and objects.

- Click the saved search drop-down  and then select a saved search. The search is performed. Configure any sorting, icon size, or other settings to suit your needs.

To set a default search in Content Explorer

The default search is performed whenever the Content Explorer window first launches. You can change the default search in the saved searches drop-down menu.

- Click the saved searches drop-down  and then click to the left of the saved search that you want to set as your default search. A check mark indicates that saved search will be the default view when you next open the Content Explorer window.



- Click the check mark beside a saved search to clear it and remove the search as the default.

TIP Perform a search for files recently modified and then save the configuration as a default search. Whenever the Content Explorer window launches, a fresh search is performed and the files you have been using most recently are displayed.

Navigate a saved search in Content Explorer

After performing a saved search, you can see the search that was performed by examining the breadcrumbs path. You can also return to a previous search or determine the folder path that was searched.

Determine a search path

- Hover your mouse over the saved search name in the breadcrumb. A tool-tip displays the folder path that was searched.

Return to a previous search

- When you are working within a saved search, click the back  button to exit the saved search and return to the previous search results.

Browse Folders with Content Explorer

You can easily browse your design files as well as the objects contained within the files.

- Double-click a folder to browse the contents of the folder.
- Double-click a file to browse the objects contained in the file.
- Double-click on a block definition to browse the block references
- Click the up  button to go to the parent folder, file, or object of your current browsing results.
  For example, if you are a browsing objects in a file, clicking the Up button returns you to the parent file. If you are currently browsing block references, clicking the Up button takes you to the block definition.

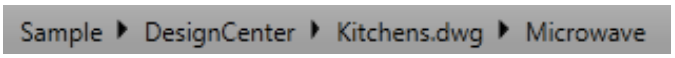
  NOTE Clicking the Up button is the same as taking one step back in the breadcrumbs path.

- Click the back  button to return to the previous browsing results.

NOTE If your browse results are not what you expected, try changing your filter settings.

Using breadcrumbs

- When browsing, a breadcrumb path records your steps.



- Click a breadcrumb to jump directly to that location.
- Click the first breadcrumb to return to the top level of the content source that you are searching or browsing.

Filter Results in Content Explorer

The filter lets you choose object types, specific dates, date ranges, and relative dates to display in search or browse results.

Filter settings persist throughout searches and browsing until they are changed or the filter is deactivated. If you do not get the search or browse results you expect, try checking your filter settings to make sure they are current.

NOTE The filter feature is disabled when using the Seek content source.

Control Whether the Filter is Turned On or Off

The Filter  icon is a toggle. When the filter is illuminated  , all search and browse results are returned based on the filter settings.

- If the filter icon is not illuminated  , click the filter to view the results based on the filter settings.

- If the filter icon is illuminated  , click the filter icon to disable it and see all objects regardless of date and time.

To filter by type in Content Explorer

1. Click the Filter  drop-down and select the Type tab.
2. Select the check boxes for the types of objects you want to display when you search or browse.

   By default, all objects are displayed when searching and browsing.
3. Once all of the settings are determined, click outside the filter menu to apply the results.

To filter by date in Content Explorer

1. Click the Filter  drop-down and select the Date tab.
2. Select whether you want to filter results by either the file's create date or the file's modified date.
3. Select a relative date from the drop-down menu.
4. Once all of the settings are determined, click outside the filter menu to apply the results.

Sort and Group Results in Content Explorer

You can control how your results are displayed through sorting and grouping options.

While a filter lets you refine which objects are displayed when you browse or search, the sort and group options let you control how those results are displayed.

Sort Results

By default, there are five columns by which you can sort results:

- Name

- Type
- Date Created
- Date Modified
- Location

Click on a column to sort the results based on that column. Click the column again to reverse the sort order.

Group Results

Sometimes even after refining a search, it might be handy to group the results based on a certain property type. You can group results by Name, Type, Date Created, or Date Modified.

When Group By is turned on, the results are collected together based on the column selected. For example, if Group By is turned on and the Type column is selected, the results are collected by the type of object—blocks, dimstyles, layers, and so on. If the Name column is selected, the results are collected by alphabetical groupings—A-H, I-P, Q-Z, 0-9, and so on.

Once you have established by which property to group your results, you can collapse or expand the groups to suit your display preferences.

To sort results

- Click a column header to sort the results by that column. Click the column again to reverse the sort order.

To display results in groups

1. Click the View Options  drop-down and select the Group By option.
2. On the Group By menu select the properties by which you want to group your browse and search results.
3. Select whether you want the results sorted in Ascending or Descending order.
4. Click outside the Group By menu to apply the settings.

To display results without grouping them

1. Click the View Options  drop-down ➤ Group By ➤ None to disable all groupings.
2. Click outside the Group By menu to apply the changes.

To expand and collapse groups

After selecting how results are grouped, you can control whether grouped results are hidden or displayed.

- Click a group heading to expand or collapse that group.
- Right-click a group header and then select Expand All or Collapse All to act on all the groups on the page.

Customize the View in Content Explorer

Whenever you search or browse, the results are displayed beneath the search field.

Click the View Options  icon to configure the icon size and whether the folders are displayed in thumbnail or detailed view. When thumbnail view is selected, you can also choose whether or not column headers and object labels are displayed. You can also choose which column headers are displayed.

To configure icon size

- Click the View Options  icon to cycle through icon sizes.

  The five icon sizes are:

  - Extra large
  - Large
  - Medium
  - Small
  - Extra small

To configure the display of folder results

1. Click the View Options  drop-down and select either Thumbnail View or Detail View from the context menu.
2. If Thumbnail View is selected, you can also choose to display the column headers or the text labels for the folders by selecting their respective check boxes.

Perform Tasks with Files in Content Explorer

Once you locate your content, you can perform different AutoCAD tasks with the files.

Task	Access
Explore	Right-click and select Explore to view the contents of the file. **NOTE** You can also double-click on the file to view its contents. This is the default action for all files.
Go to Folder	Right-click on a file and select Go to Folder to go directly to the folder containing that file.
Open Drawing	Right-click and select Open Drawing to open the file in the current AutoCAD session. You can also drag and drop a file from the display window into the AutoCAD session to open it provided no other files are currently open.
Open and Find Text	Right-click and select Open and Find text to open the drawing and locate any text within. **NOTE** If the option is grayed out then there is no text in the drawing file.

Task	Access
	Generally, the command will automatically highlight and zoom in to the text, except in these cases: ■ The text is a file-level property so there is nothing to highlight on the screen. ■ The text is an object definition property that has not been instantiated in the file. For example, a block attribute for a block that has not been inserted would not be higlighted. **NOTE** You can instead examine the properties on the block to see the text.
Insert	Right-click on the file and select Insert to insert the file into the current AutoCAD document. You can also drag and drop the file from the display window into the current document window to insert it.
Attach	Right-click on the file and select Attach to attach the file as an Xref to the current AutoCAD document.
Properties	Right-click on the file and select Properties to view a read-only list of the properties stored in the index for the file.

Perform Tasks with Objects in Content Explorer

Once you locate an object, you can perform different AutoCAD tasks with the objects.

Task	Access
Explore	Double-click on a block definition or right-click and select Explore to see instances of the block definition.

Task	Access
NOTE This feature is available only for block references.	
Go to Folder	Right-click on an object and select Go to Folder to go directly to the folder containing the object. Right-click on a block definition and select Go to Folder to go directly to the parent file containing the block definition.
Open Drawing	Right-click on an object and select Open to Open the drawing containing the selected object.
Open and Find Text	Right-click and select Open and Find text to open the drawing and locate any text within. **NOTE** If the option is grayed out then there is no text in the drawing file. Generally, the command will automatically highlight and zoom in to the text, except in these cases: ■ The text is a file-level property so there is nothing to highlight on the screen. ■ The text is an object definition property that has not been instantiated in the file. For example, a block attribute for a block that has not been inserted would not be higlighted. **NOTE** You can instead examine the properties on the block to see the text.
Open and Zoom	Right-click on the object and select Open and Zoom to open the drawing in the current AutoCAD session and highlight the selected object.

Task	Access
Insert	Right-click on the object and select Insert to insert the object into the current AutoCAD document. You can also drag and drop the object from the display window into the current document window to insert it.
Properties	Right-click on the object and select Properties to view a read-only list of the properties stored in the index for the object.

Streamline Searches with User-Created Properties in Content Explorer

Searches can be refined based on the properties associated with drawing files.

By adding custom properties to a drawing file, you can create unique tags for searching and grouping.

For example, if you have several furniture files which you regularly use when designing, you can add a property to each of the files named FURNITURE TYPE and give each a suitable value, such as CHAIR. Then when you perform a search for files with FURNITURE TYPE as a property and the value of CHAIR, you will retrieve only the files that meet this criterium.

Understand the Drawing Properties Dialog Box in AutoCAD

The AutoCAD properties dialog box contains four tabs that organize the properties.

Here is a description of each tab:

Tab	Description
General	The General tab contains the basic system level properties.
Summary	The Summary tab contains file properties that can be filled in by the user. These properties include the title, subject, author, keywords, and comments about the file. If

Tab	Description
	the file is contained in a watched folder, when the file is saved, these properties are indexed and can be searched using Content Explorer. **TIP** The property values for the properties contained in the Summary tab are searched automatically with a general search string. Fill in these values as soon as possible so that you can start refining your searches using the property:value search feature.
Statistics	The Statistics tab displays the created and modified dates for the file. It also displays the name of the last person to save the file as well as any revision information related to the file.
Custom	The Custom tab displays all user-created properties for the file. Click Add to create a property:value pair and then Click Save. If the file is contained in a watched folder, the new properties are indexed and can be searched using Content Explorer.

You can create custom property values for different drawing files. These property values can be used to locate files in watched folders when you search with Content Explorer.

To add properties to a drawing file

1. From the AutoCAD application menu, select Drawing Utilities ➤ Drawing Properties.
2. Click the Summary tab and enter the desired values in each of the fields.
3. Click the Custom tab.
4. On the Custom tab, click Add.
5. Enter a new property and the desired value for this file.
6. Click Save to add the property and designated value to the file.

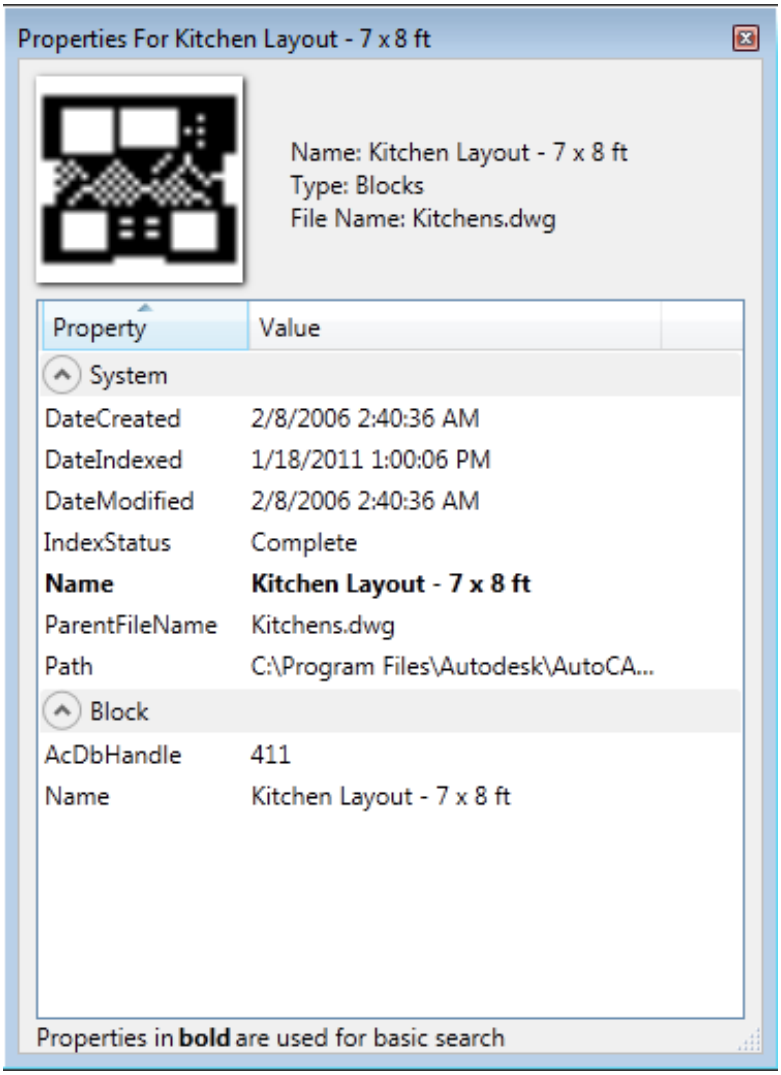
If the file is already contained in a watched folder, the new property is automatically indexed. The property and value can be searched using Content Explorer.

View Properties on an Indexed File or Object

You can view which properties have been indexed on a file or an object. Knowing which properties have been indexed allows you to create stronger search strings.

1. Right-click on the object or file for which you want to view the properties and select Properties.

   A window displays all of the properties that have been indexed, including the default properties and any user-created properties.



NOTE Properties in bold are default properties.

TIP Advanced users can add custom properties to the basic search list by accessing the Connect.Service.exe.config file and following the instructions under the <BasicSearchProperties> heading.

2 Select another object or file to view its properties as well.

The properties window automatically updates to display the properties that have been indexed on the selected file or object.

Use Saved Searches with User-Created Properties in Content Explorer

When the Saved Searches feature is used in conjunction with properties and searching, you can create fast access to the files you use most often.

For example, if you frequently use furniture files when designing, you can add a FURNITURE TYPE property with suitable values, such as SEATING, to your furniture files. Then you can create a search that locates all files with the property FURNITURE TYPE and a value of SEATING, and then save that search for faster recall the next time you need to locate seating options for your design.

You can further refine the search by grouping and filtering the results to suit your working preferences. For example, FURNITURE TYPE results for SEATING can be grouped in the display window based on whether they have been modified recently or not.

See also:

- Advanced Searching with Content Explorer on page 126
- Manage Saved Searches in Content Explorer on page 128

Troubleshooting Content Explorer

The troubleshooting section provides useful tips for resolving any issues with Content Explorer.

Content Service is not installed

There are two components that help you locate your files. Content Explorer is the interface that allows you to browse and search watched folders. Content Service is the service that indexes the files and enables the searching. When AutoCAD is first installed, the Autodesk Content Explorer feature is automatically included in the installation. At the same time, the Content Service is also installed, but since the service runs independent of AutoCAD, it shows up as a separate program on the Add and Remove programs list.

If you accidentally uninstall the service, the Content Explorer interface will no longer work within AutoCAD.

To reinstall Content Service

1. Launch the AutoCAD installation wizard.
2. Click the Tools and Utilities button.
3. On the Configure Settings screen, select one of these options under Autodesk Content Service:
   - For Personal Access - Select this option if you intend to use AutoCAD with Content Explorer on your personal machine.
   - For Public Access - Select this option if you want to index files on the local machine so that all Content Explorer users can connect to the machine and explore and search files.

     NOTE See the Content Service help file for more information on planning your installation.
4. Click Install to start the installation.

You are receiving a Cannot Perform Operation error

If you receive an error in the Content Explorer window stating that you cannot perform an operation, it may be due to one of the following reasons:

- If you are trying to access a network conetnt source or Autodesk Seek, you may not be able to connect because your internet connection is lost or has been disrupted. Check your internet connection to make sure you are still connected.
- If you are trying to access a network content source or Autodesk Seek, and you DO have a valid internet connection:
  - You might not be able to connect due to a DNS error. You can verify if this is the problem by trying to connect to the network computer using Windows Explorer (for example, try connecting to \\COMPUTERNAME). If you can't connect in this way, then there may be a permissions, DNS, or network problem in general. Always make sure that you can access the network computer using Windows Explorer first, because if you cannot access the network computer using Windows Explorer, you will not be able to access the network computer with Content Explorer either.

- The Content Service may have stopped running. Check to make sure that it is still running by typing Services in the Search box of the Windows Start menu. In the Services window, locate Autodesk Content Service in the list of services and make sure that the status is Started and the Startup Type is Automatic.
- When trying to connect to a network computer, you may encounter firewall issues. When installed, an inbound rule is created for the Windows Firewall that enables Autodesk Content Service on port 50248 using TCP protocol (this applies to the Domain profile). You can check these settings in the list of Inbound Rules for Windows Firewall.
  1. Enter Windows Firewall in the Search box on the Windows Start menu.
  2. When the Windows Firewall window appears, click Inbound Rules in the left pane, and then locate Autodesk Content Service in the list of inbound rules.
  3. Right-click on Autodesk Content Service and select Properties.
  4. Select the Advanced tab.
  5. Select the Protocols and Ports tab to see the settings for Autodesk Content Service.
     Make sure that the Autodesck Content Service is on port 50248 and is using TCP protocol. Different settings could cause connection issues.

You cannot open or insert a file from a network machine in Content Explorer

Autodesk Content Service runs as a local system on the computer where it is installed. It will index any files that it has access to within the specified watched folders. Searching is based on the permissions of the local system--even if you are searching from a remote client.

When searching, any user will be able to see any of the files that are indexed. If a user cannot open and insert any of these files, check that the Windows permissions for those files allow local system access.

Text and objects within an xref are not searchable

If your files contain external references, the text and objects within those references won't be extracted for the parent file. Therefore, text and objects within an xref won't be searchable within the content of the parent file.

You cannot find the Plug-ins tab on the ribbon or the Explorer button does not show up on the Plug-ins tab

If the Plug-ins tab does not show up on the ribbon or the Explorer button does not show up on the Plug-ins tab of the ribbon, one reason might be that Content Explorer is not loaded as an add-in.

To manually reload Content Explorer as an add-in

1. Go to the AutoCAD command line.
2. Type CUILOAD.
3. If CONTENTEXPLORER is in the list, click the Unload button to unload it.
4. Browse for the contentexplorer.cuix file.
5. Once you located the file, click the Load button to re-load it.

   The Plug-ins tab should now appear on the ribbon with the Explorer button.

   NOTE You can always start Content Explorer from the command line by entering CONTENTEXPLORER.

Sheet Set Manager dialog opens every time you open a file in Content Explorer

If the Sheet Set Manager dialog opens every time you open a file in Content Explorer, it is because of the SSMAUTOOPEN system variable setting. Change the setting to suppress the dialog. Refer to SSMAUTOOPEN for more information.

There is no ribbon access in AutoCAD ecscad for Content Explorer

AutoCAD ecscad does not support the ribbon interface. Use the following commands to open and close the Content Explorer window.

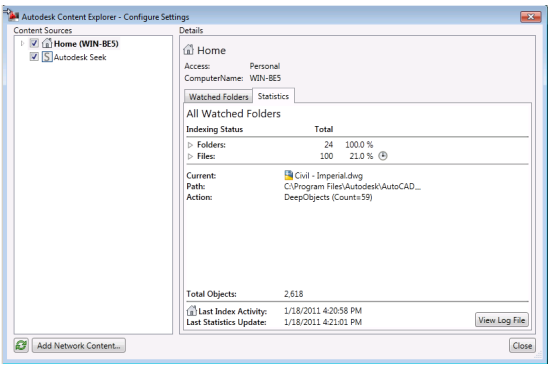
- To open Content Explorer, enter CONTENTEXPLORER on the command line.
- To close Content Explorer, enter CONTENTEXPLORERCLOSE on the command line.

Indexing process is producing errors

There are various reasons why the indexing process might encounter an issue. If you suspect that there were issues when watched folders were being indexed, you can review the indexing statistics to find out more about the errors.

To review the indexing statistics in Content Explorer

1. In Content Explorer, click Configure Settings.
2. On the Configure Settings dialog, select either your local computer or a network content source and click the Statistics tab.



3. Expand the indexing statistics for a file or folder to see a list of known issues.
4. If the indexing service stalled without any current actions, remove the watched folder and re-add it for indexing.

   This action will start the indexing process over and will resolve most issues.

NOTE The Statistics tab is also available in the Autodesk Content Service Administration Console. Refer to the Autodesk Content Service Administration Console help for more information.

Customize the Drawing Environment

5

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 grip editing behavior.

Many of the settings are available from shortcut menus and the Options dialog box.

Some user interface elements, such as the presence and location of menu items and palettes, can be specified and saved using the Customize User Interface Editor.

Some settings affect how you work in the drawing area:

- **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, application menu, 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.
- **UCS Icon Style, Size, and Color (UCS Icon dialog box).** You can control the appearance of the UCS icon in model space and paper space.
- **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 application menu, 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.

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.

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.

To display hidden message dialog boxes

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 box 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 window.

CLEANSCREENOFF

Restores the state of the display before CLEANSCREENON was used.

DRAGMODE

Controls the way dragged objects are displayed.

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

APPLYGLOBALOPACITIES

Applies transparency settings to all palettes.

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 display of objects being dragged.

DRAGP1

When hardware acceleration is used, controls how many vectors the system draws when you drag objects within a 2D viewport before it checks for a new input sample from the mouse.

DRAGP2

When software acceleration is used, controls how many vectors the system draws when you drag objects within a 2D viewport before it checks for a new input sample from the mouse.

EXTNAMES

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

GLOBALOPACITY

Controls transparency level for all palettes.

GRIPCOLOR

Controls the color of unselected grips.

GRIPHOT

Controls the color of selected grips.

GRIPS

Controls the display of grips on selected objects.

HELPPREFIX

Sets the file path for the Help system.

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 command history are written to a log file.

LOGFILENAME

Specifies the path and name of the command history log file for the current drawing.

LOGFILEPATH

Specifies the path for the command history 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 box 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 for object selection.

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.

ROLLOVEROPACITY

Controls the transparency of a palette while the cursor moves over the palette.

ROLLOVERTIPS

Controls the display of rollover tooltips when the cursor hovers over an object.

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.

SHORTCUTMENU

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

SHORTCUTMENUDURATION

Specifies how long the right button on a pointing device must be pressed to display a shortcut menu in the drawing area.

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

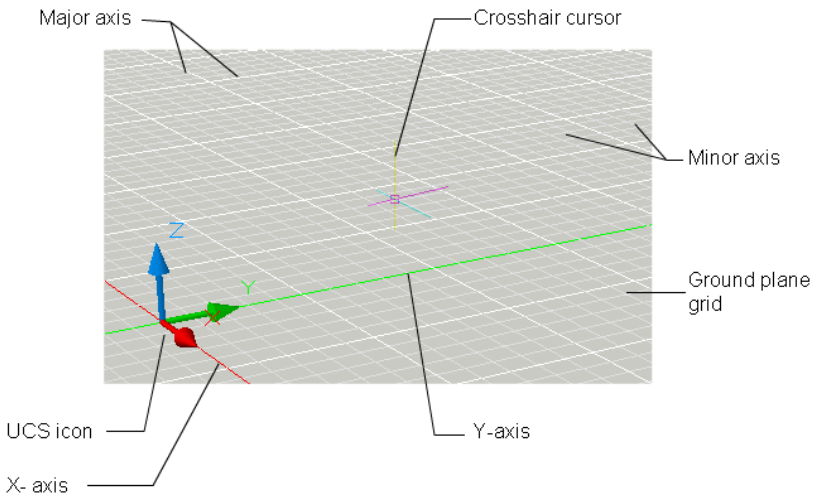
Turns xref demand-loading on and off, and controls whether it opens the referenced drawing or a copy.

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 display of grips on selected objects.

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 command history are written to a log file.

LOGFILENAME

Specifies the path and name of the command history log file for the current drawing.

LOGFILEPATH

Specifies the path for the command history 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 box 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 for object selection.

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 when the cursor hovers over an object.

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.

Switch Between Model and Named Layouts

You can control how you change between the Model and one or more named 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 display of grips on selected objects.

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 command history are written to a log file.

LOGFILENAME

Specifies the path and name of the command history log file for the current drawing.

LOGFILEPATH

Specifies the path for the command history 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 box 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 for object selection.

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 when the cursor hovers over an object.

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.

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 Display tab (Options dialog box).

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.

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 display of grips on selected objects.

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 command history are written to a log file.

LOGFILENAME

Specifies the path and name of the command history log file for the current drawing.

LOGFILEPATH

Specifies the path for the command history 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 box 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 for object selection.

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 when the cursor hovers over an object.

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

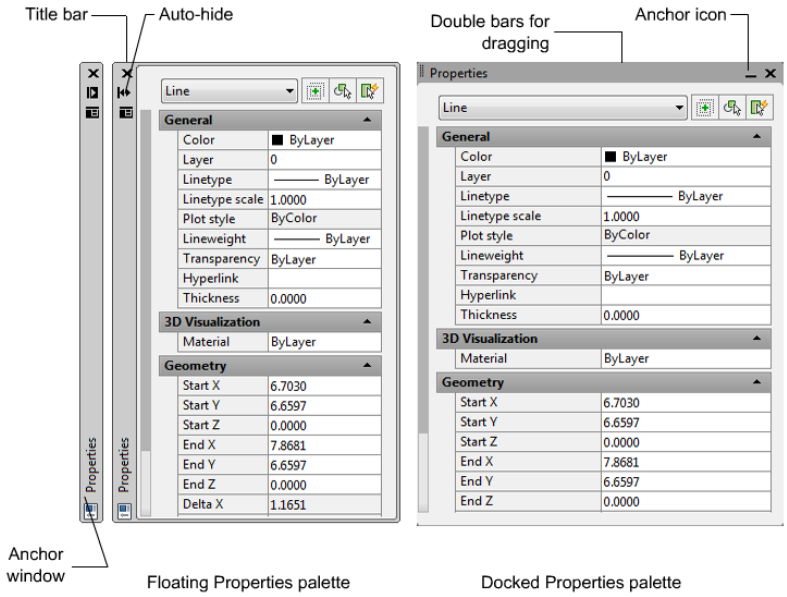
Turns xref demand-loading on and off, and controls whether it opens the referenced drawing or a copy.

Specify the Behavior of Dockable Windows

Windows such as the ribbon, 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.



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 all currently displayed palettes, including the command window.

LOGFILEOFF

Closes the command history log file opened by LOGFILEON.

LOGFILEON

Writes the contents of the command history 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 display of grips on selected objects.

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 command history are written to a log file.

LOGFILENAME

Specifies the path and name of the command history log file for the current drawing.

LOGFILEPATH

Specifies the path for the command history 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 box 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 for object selection.

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 when the cursor hovers over an object.

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.

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.

Quick Reference

Commands

CLEANSCREENON

Clears the screen of toolbars and dockable windows, excluding the command window.

CLEANSCREENOFF

Restores the state of the display before CLEANSCREENON was used.

DRAGMODE

Controls the way dragged objects are displayed.

HIDEPALETTES

Hides all currently displayed palettes, including the command window.

LOGFILEOFF

Closes the command history log file opened by LOGFILEON.

LOGFILEON

Writes the contents of the command history 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.

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

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

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

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Controls the graphics refresh rate.

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

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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 command history are written to a log file.

LOGFILENAME

Specifies the path and name of the command history log file for the current drawing.

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Specifies the path for the command history log files for all drawings in a session.

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Sets the application for editing multiline text objects.

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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 box 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 for object selection.

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 when the cursor hovers over an object.

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.

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 25.

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.

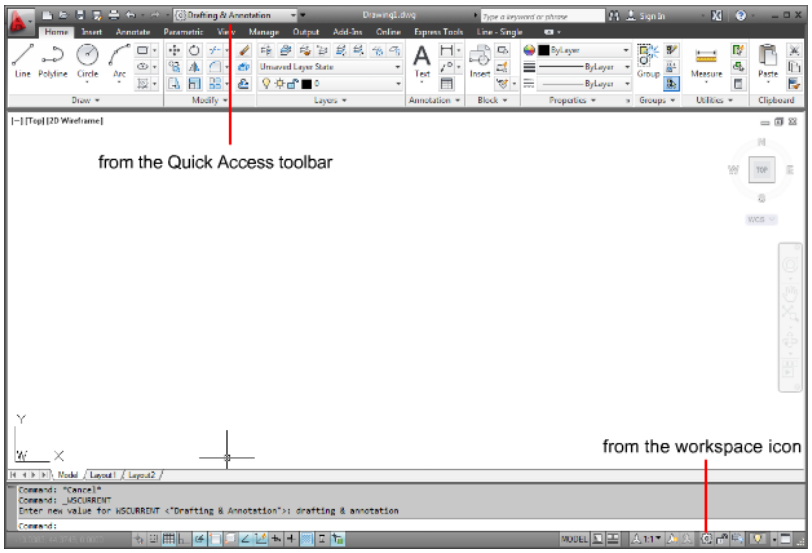
Interface items that you do not need for 3D modeling are hidden, maximizing the screen area available for your work.

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.

NOTE Saved workspaces are also accessible from the Quick Access toolbar.



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 Quick Access toolbar, 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 2012\<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 193.

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.

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.

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.

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

WORKSPACELABEL

Controls whether to display the name of the current workspace in the status bar.

WSAUTOSAVE

Saves changes you made to a workspace when you switch to another work space.

WSCURRENT

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

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*.

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.

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.

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.

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 *acad2010.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 the name of a profile that is listed in the Options dialog box or the file name of an exported profile (ARG) file. 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/publishing	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 2012* with the drawing template *arch1.dwt*, restores a named view PLAN1, and executes a script file *startup.scr*.

"d:\AutoCAD 2012*acad.exe*"/t "d:\AutoCAD 2012\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 2012*acad.exe*" **/t** "**d:**\AutoCAD 2012**template\arch1**" **/v** "**plan1**" **/b** "**startup**"

3 Click OK.

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

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.

Migrate Custom Settings and Files

Migrating from an older release of AutoCAD allows you to use your custom settings and files in the latest release.

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 migrate and transfer your custom settings and files to the latest release.

See also:

- Migrate Custom Settings and Files from Previous Releases in the *Customization Guide*
- Basic Customization in the *Customization Guide*
- User Interface Customization in the *Customization Guide*

Part 3: Start and Save Drawings

Start a Drawing 6

All drawings start from either a default drawing template file or a custom drawing template file that you create. Drawing template files store default settings, styles, and additional data.

Overview of Starting a New Drawing

Before you start to draw, you need to decide what system of *drawing units* that you will use in the drawing, and then choose a *drawing template file* appropriate for those drawing units.

Choose Drawing Units

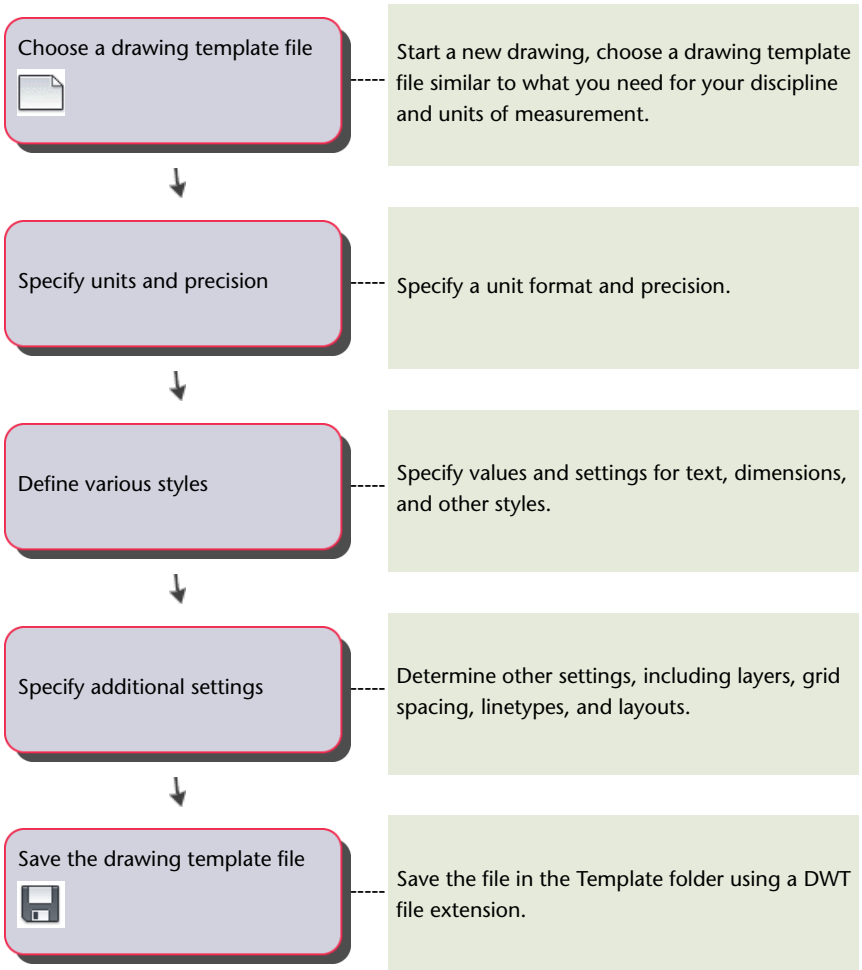
A drawing unit can equal one inch, one millimeter, or any other system of measurement. For more information about drawing units, see Determine the Units of Measurement on page 207.

Choose a Drawing Template File

When you start a new drawing, AutoCAD accesses a drawing template file to determine many default settings such as unit precision, dimension styles, layer names, a title block, and other settings. Many of the settings are based on whether the drawing template file is intended for use with a drawing created in inches, feet, millimeters,centimeters, or other unit of measurement.

Customize a Drawing Template File

By customizing your own drawing template file, you save yourself a lot of work changing settings, and you also ensure that the settings are standardized.



You can create several drawing template files for different projects, and you can choose one when you click New.

Quick Reference

Commands

NEW

Creates a new drawing.

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.

Specify Units and Unit Formats

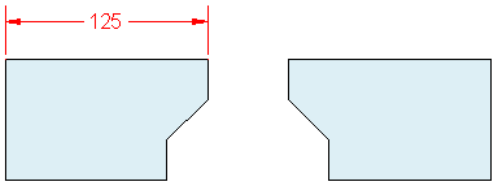
Before you start to draw, you decide on the units of measurement to be used in the drawing, and set the format, precision, and other conventions to be used in coordinates and distances.

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, the objects in the illustration might represent two buildings that are each 125 feet long, or a section of a mechanical part that is measured in millimeters.



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 1567

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.

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.

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.

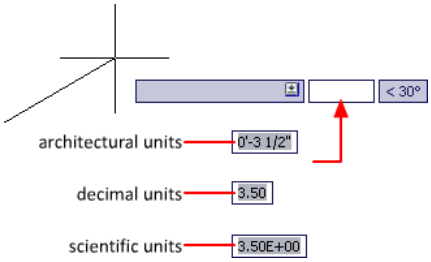
Set the Unit Format Conventions

You can set the format and the number of decimal places to be used when you enter and display linear and angular units.

Set Linear Units

You can choose from several common conventions to represent the format and the precision of linear distances and coordinates displayed in the Properties palette, dynamic input, the status bar, and other locations.

For example, here are three variations of dynamic input.



Set Angular Units

You can specify that positive values of angles are measured either clockwise or counterclockwise, and the direction of angle 0 (usually East or North). You can enter angles using grads, radians, or surveyor's units or using degrees, minutes, and seconds.

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 the relative coordinates 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**.

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.

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.

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.

Use a Drawing Template File

A drawing template file provides consistency in the drawings that you create by maintaining your standard styles and settings.

Select a Drawing Template File

A set of drawing template files is installed with AutoCAD. Many of them are provided either for imperial or for metric units, and some are optimized for 3D modeling. All drawing template files have a *.dwt* file extension.

While these drawing templates provide a quick way to start a new drawing, it is best to create drawing templates specific to your company and the type of drawings you create.

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 format and precision on page 207
- Title blocks and borders on page 917
- Layer names on page 516
- Snap and Grid spacing on page 635
- Text styles on page 1469
- Dimension styles on page 1530
- Multileader styles on page 1451
- Table styles on page 1511
- Linetypes on page 563
- Lineweights on page 574
- Layouts on page 433
- Page setups on page 1635

By default, drawing template files are stored in the *template* folder, where they are easily accessible. You can use the Options dialog box to set a default for both the template folder and the drawing template file.

See also:

- Use a Hyperlink to Start a New Drawing on page 2086

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.

   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.

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. Erase all of the objects in the drawing that you do not want to keep.
4. 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.
5. In the Save Drawing As dialog box, File Name text box, enter a name for the drawing template.
6. Click Save.
7. Enter a description for the drawing template.
8. Click OK.

   The new template is saved in the *template* folder.

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.

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 Finder

Finder can be used to location drawing files. 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 Access Content with DesignCenter on page 87.

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 491.

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 1458.

To display drawing properties for the active drawing

1. Click the Application button, and click Drawing Utilities ➤ Drawing Properties. 
2. In the Drawing Properties dialog box, click tabs to view the different types of information.

To define drawing properties

1. Click the Application button, and click Drawing Utilities ➤ 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.

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.

Specify the Geographic Location of a Drawing

You can specify the geographic location, orientation, and elevation of the objects in a drawing.

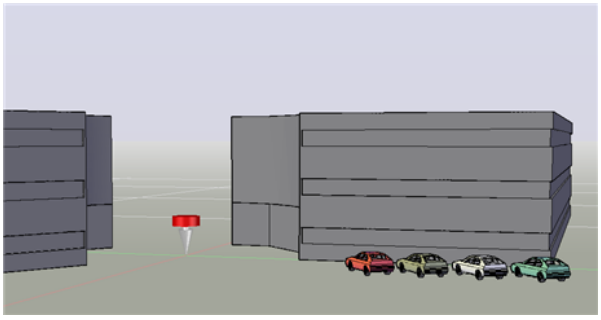
This information is useful for sun studies, environment analysis, exporting to AutoCAD Map 3D, and working with Google Earth.

Geographic location information can be specified in one of the following ways:

- Enter the latitude, longitude, and altitude manually

- Import a KML or KMZ file with the geographic information
- Import a location from Google Earth

When you specify the geographic location of a drawing, a geographic marker is created. The geographic marker is a visual representation of the location information, and is created at a specified point on the drawing.



This marker cannot be selected, but you can control whether it is displayed using the GEOMARKERVISIBILITY system variable.

You can display the latitude and longitude of the cursor in the coordinate display area of the status bar.

To specify the geographic location of a drawing

1. Click Render tab ➤ Sun & Location panel ➤ Set Location. 
2. Click Enter the Location Values.
3. (Optional) Select the latitude and longitude format, or click Use Map to specify the nearest city and time zone.
4. Specify the *X*, *Y*, and *Z* coordinates in the drawing corresponding to the geographic data.

   The north direction angle is calculated when you select a point with reference to the geographic location.
5. Click OK.

To import a kml or kmz file into a 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 a KML or KMZ file references multiple locations, only the first location is used. Click Close if the Multiple Locations Found dialog box is displayed.

4. Click or specify the coordinates for the location in the World Coordinate System (WCS) *X*, *Y*, *Z* format.
5. Click to specify the north direction.

   A geographic marker is inserted at the specified location.

To import a geographic location from Google Earth

1. Start Google Earth and select a location.
2. Click Render tab ➤ Sun & Location panel ➤ Set Location. 
3. Click Import the current location from Google Earth.
4. Click Continue.
5. Click or specify the coordinates for the location in the World Coordinate System (WCS) *X*, *Y*, *Z* format.
6. Click to specify the north direction.

   The geographic marker is created at the specified point on the drawing.

To edit or delete a geographic marker

1. Click Render tab ➤ Sun & Location panel ➤ Set Location. 
2. Do *one* of the following:

- Click Edit Current Geographic Location.
- Click Remove 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 DWG files 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 7

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 applications. In addition, you can choose from several alternative methods.

To open a drawing, you can

- Use Open on the File menu 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.
- Use Open on the Quick Access toolbar to display the Select File 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 files, see ■ Resolve Missing External References on page 1808 ■ Attach Raster Image Files on page 1916 ■ Attach Files as Underlays on page 1884

Missing Reference Types	Description
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 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 on page 2109.

Work with Large Objects

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 .

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 247.

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 start AutoCAD in the folder specified in the "Start in" property of the shortcut. Set REMEMBERFOLDERS to 0 and use the revised shortcut to start AutoCAD from the folder specified in the "Start in" property.

See also:

- Overview of DesignCenter on page 84
- Work with Sheets in a Sheet Set on page 471
- Open and Save Drawing Files from the Internet on page 2088
- Overview of Using Markups for Design Review on page 2125

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.

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.

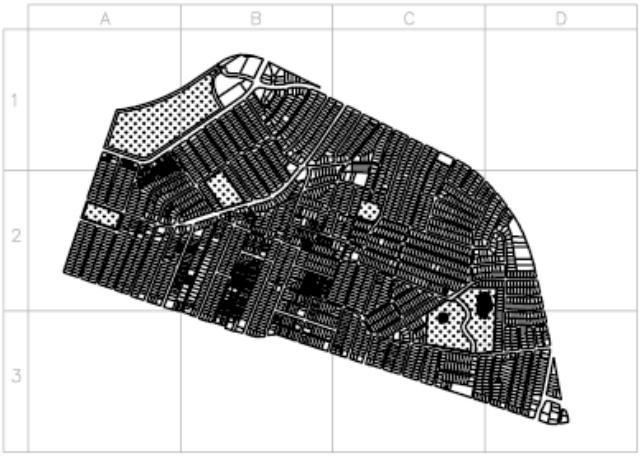
Open Part of a Large Drawing (Partial Load)

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.

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.

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.

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 230.

- **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 234.

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

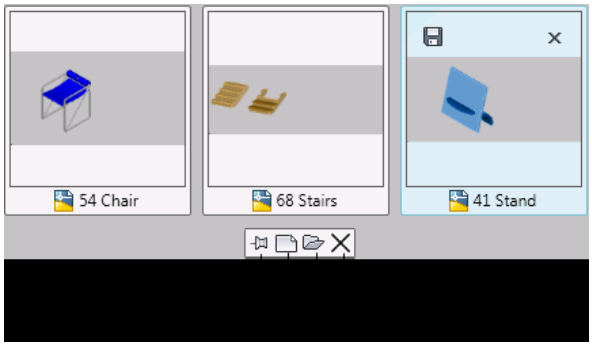
Controls the display of the application and drawing status bars.

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 Quick View Toolbar

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.

Use the Quick View Drawings Tool

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.

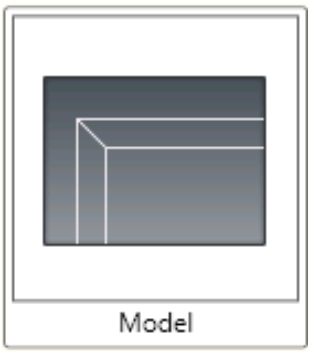
You can access more options for layouts from the Quick View layout image. For more information, see Switch Between Layouts in the Current Drawing on page 234.

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 preview and switch between open drawings

1. Click  the status bar.
   A row of Quick View images is displayed at the bottom of the program.
2. Click the Quick View image you want to work on.

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 program.
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.

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 program.
2. Press Ctrl + scrollwheel over a Quick View image to resize.

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 for named views, drawings, and layouts.

System Variables

QVDRAWINGPIN

Controls the default display state of preview images of drawings.

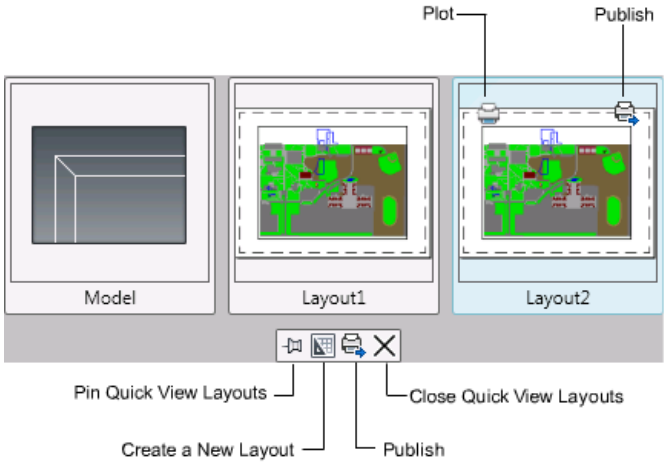
UPDATETHUMBNAIL

Controls updating of the thumbnail previews for views and layouts.

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.

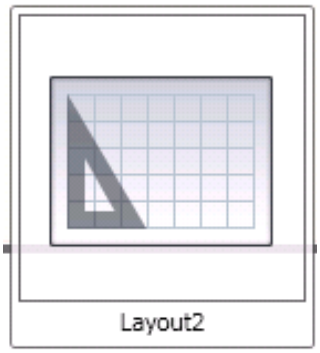
If a row of Quick View images extends beyond the application display area, scroll arrows display on the left or right side of the row. You can scroll to view the other images.

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 update a thumbnail image

NOTE Verify that **updatethumbnail** is not set to **0**

1. At the Command prompt, enter **updatethumbsnow**.
2. On the status bar, click  .

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 for named views, drawings, and layouts.

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 for views and layouts.

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.

Quick Reference

Commands

COPYBASE

Copies selected objects to the Clipboard along with a specified base point.

PASTEORIG

Pastes objects from the Clipboard into the current drawing using the original coordinates.

OPTIONS

Customizes the program settings.

TASKBAR

Controls whether multiple open drawings are displayed separately or grouped on the Windows taskbar.

Save a Drawing

You save drawing files for later use just as you do with other 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 in the Save Drawing As dialog box.

Save with Visual Fidelity for Annotative Objects

When working with annotative 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 drawing or DXF file formats.

Annotative objects may have multiple scale representation. When visual fidelity is on, annotative objects are decomposed and scale representations are saved (in an *anonymous block*) 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 layout. More annotation objects may be displayed on the Model layout 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 the operating system, 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 because the product is Unicode-compliant.

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 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 on page 1960
- Work with Drawings in Earlier Releases on page 1954
- Export Drawings to Other File Formats on page 1938
- Create Drawing Files for Use as Blocks on page 931
- Add Identifying Information to Drawings on page 214
- Create and Restore Backup Files on page 250
- Share Drawing Files Internationally on page 2092
- Open and Save Drawing Files from the Internet on page 2088

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.

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.

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.

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.

Quick Reference

Commands

BLOCK

Creates a block definition from selected objects.

OPTIONS

Customizes the program settings.

QSAVE

Saves the current drawing using the specified default file format.

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

Saves selected objects or converts a block to a specified 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 214
- Access Content with DesignCenter on page 87
- Work with Sheets in a Sheet Set on page 471

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.

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 (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 (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 the operating system 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 (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.

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

8

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. A recovery notification is displayed while opening drawing files that are damaged and need recovery. You can

- 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 current drawing.
- RECOVERAUTO. Controls the display of recovery notifications before or after opening a damaged 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).

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.

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.

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.

RECOVERAUTO

Controls the display of recovery notifications before or after opening a damaged drawing file.

REPORTERROR

Controls whether an error report can be sent to Autodesk if the program closes unexpectedly.

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 Open and Save tab (Options dialog box), 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 252

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.

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.

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.

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.

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.

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.

Maintain Standards in Drawings

9

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

CHECKSTANDARDS

Checks the current drawing for standards violations.

STANDARDS

Manages the association of standards files with drawings.

STANDARDSVIOLATION

Specifies whether a user is notified of standards violations in the current drawing when a nonstandard object is created or modified.

Batch Standards Checker

Audits a set of drawings for standards violations.

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.

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.

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.

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.

Quick Reference

CHECKSTANDARDS

Checks the current drawing for standards violations.

STANDARDS

Manages the association of standards files with drawings.

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.

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.

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.

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.

To start the Batch Standards Checker

- Click Start menu (Windows) ➤ All Programs ➤ Autodesk ➤ AutoCAD ➤ 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. Click Manage tab ➤ CAD Standards panel ➤ Check.
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. Click Manage tab ➤ CAD Standards panel ➤ Check.
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. Click Manage tab ➤ CAD Standards panel ➤ Check.
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 Click Manage tab ➤ CAD Standards panel ➤ Check. 

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 Click Manage tab ➤ CAD Standards panel ➤ Check. 

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 Click Manage tab ➤ CAD Standards panel ➤ Check. 

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 Click Manage tab ➤ CAD Standards panel ➤ Check. 

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

CHECKSTANDARDS

Checks the current drawing for standards violations.

STANDARDS

Manages the association of standards files with drawings.

STANDARDSVIOLATION

Specifies whether a user is notified of standards violations in the current drawing when a nonstandard object is created or modified.

Batch Standards Checker

Audits a set of drawings for standards violations.

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.

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.

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.

Quick Reference

LAYTRANS

Translates the layers in the current drawing to specified layer standards.

Part 4: Control the Drawing Views

Change Views

10

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

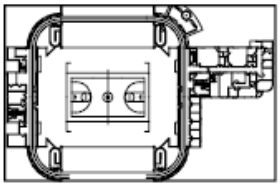
You can pan to reposition the view in the drawing area or zoom to change magnification.

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.

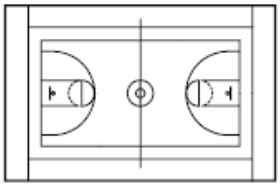
You can change the magnification of a view by zooming in and out, which is similar to zooming in and out with a camera. ZOOM does not change the absolute size of objects in the drawing; it changes only the magnification of the view.

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.



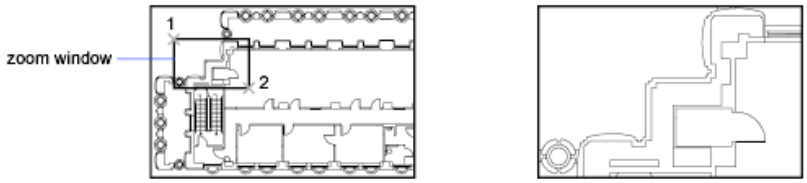
zoomed out



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.

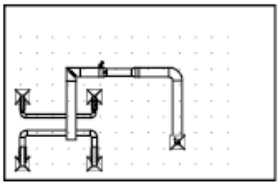
Zoom to Magnify One or More Objects

ZOOM Objects displays a view with the largest possible magnification that includes all of the objects you selected.

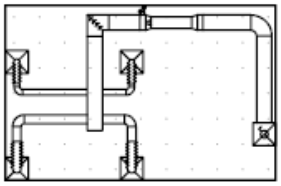
Zoom to View All Objects in the Drawing

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.

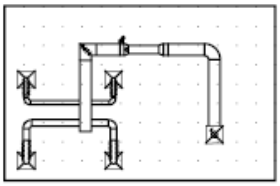
ZOOM All displays either the user-defined grid limits or the drawing extents, whichever view is larger.



current view



zoomed to extents



zoomed to show entire drawing (all)

See also:

- Scale Views in Layout Viewports on page 450
- Pointing Devices
- Toolbars on page 40

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.

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 ➤ Zoom drop-down ➤ 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.

To zoom in to an area by specifying its boundaries

1 Click View tab ➤ Navigate panel ➤ Zoom drop-down ➤ 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.

To display the drawing extents by zooming

■ Click View tab ➤ Navigate panel ➤ Zoom drop-down ➤ 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.

To display the area of the grid limits by zooming

■ Click View tab ➤ Navigate panel ➤ Zoom drop-down ➤ 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.

To restore the previous view

■ Click View tab ➤ Navigate panel ➤ Zoom drop-down ➤ Previous.



Zoom Previous restores only the view magnification and position, not the previous content of an edited drawing.

Quick Reference

Commands

PAN

Moves the view planar to the screen.

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.

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.

A named view created with the VIEW command consists of a specific magnification, position, and orientation. In each drawing session, you can restore up to 10 previous views displayed in each viewport using ZOOM Previous.

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 or a specific named layout)
- 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 can use named views to do the following:

- Restore a view that you use frequently while working in model space.
- Restore a view on a layout that is zoomed into an area of interest on the layout.
- With multiple model or layout viewports, restore a different view in each one.

To restore the previous view

- Click View tab ➤ Navigate panel ➤ Zoom drop-down ➤ Previous.



Zoom Previous restores only the view magnification and position, not the previous content of an edited drawing.

To display the previous view during PAN Realtime and ZOOM Realtime

- Click View tab ➤ Navigate panel ➤ Zoom drop-down ➤ Previous 

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 ➤ View Manager. 
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.

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 ➤ View Manager. 
3. In the View Manager, Views list, select the view that you want to restore.
4. Click Set Current. Click OK.

To rename a view

1. Click View tab ➤ Views panel ➤ View Manager. 
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.

To change the properties of a view

1. Click View tab ➤ Views panel ➤ View Manager. 
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.

To delete a named view

1. Click View tab ➤ Views panel ➤ View Manager. 
2. In the View Manager, click a view, and then click Delete.
3. Click OK.

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.

Quick Reference

Commands

CAMERA

Sets a camera and target location to create and save a 3D perspective view of objects.

VIEW

Saves and restores named model space 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

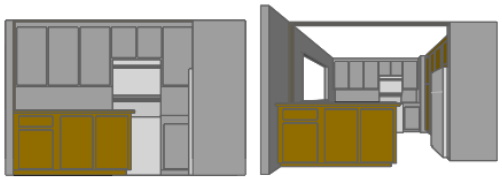
You can view both parallel and perspective projection of a 3D model.

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

Determines whether input to the DVIEW and VPOINT commands is relative to the WCS (default) or the current UCS.

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.

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.

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

Determines whether input to the DVIEW and VPOINT commands is relative to the WCS (default) or the current UCS.

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.

Quick Reference

Commands

DVIEW

Defines parallel projection or perspective views by using a camera and target.

PLAN

Displays an orthographic view of the XY plane 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

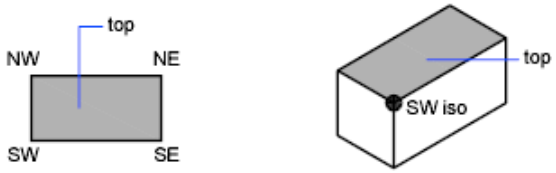
Determines whether input to the DVIEW and VPOINT commands is relative to the WCS (default) or the current UCS.

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 ➤ View Manager.
- Select a preset view (Top, Bottom, Left, and so on).

Quick Reference

Commands

VIEW

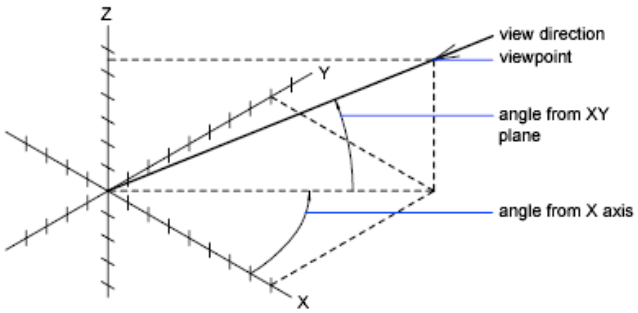
Saves and restores named model space views, layout views, and preset views.

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.

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.

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.

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.

Quick Reference

Commands

VIEW

Saves and restores named model space views, layout views, and preset views.

VPOINT

Sets the viewing direction for a 3D visualization of the drawing.

System Variables

WORLDVIEW

Determines whether input to the DVIEW and VPOINT commands is relative to the WCS (default) or the current UCS.

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.

Quick Reference

Commands

DVIEW

Defines parallel projection or perspective views by using a camera and target.

PLAN

Displays an orthographic view of the XY plane 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

Determines whether input to the DVIEW and VPOINT commands is relative to the WCS (default) or the current UCS.

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

Visual styles control the display of edges and shading a viewport.

Control the effect of a visual style by changing its properties. When you apply a visual style or change its settings, the associated viewport is automatically updated to reflect those changes.

The Visual Styles Manager displays all 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.

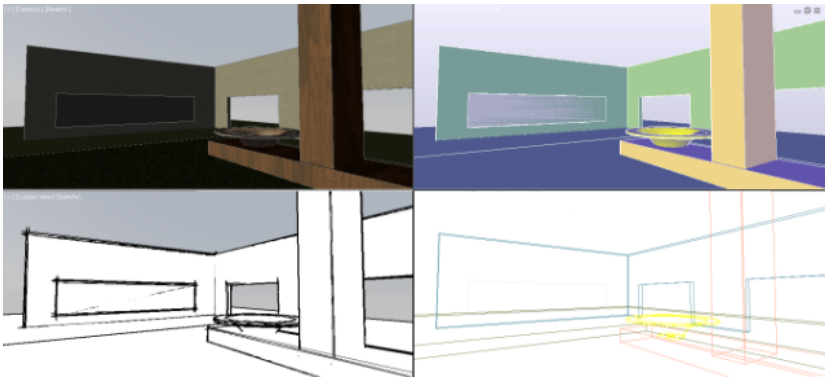
From the ribbon on page 25, you can change some frequently used settings or open the Visual Styles Manager.

The following predefined visual styles are supplied with the product:

- **2D Wireframe.** Displays objects using lines and curves to represent the boundaries.

  NOTE Raster images, OLE objects, linetypes, and lineweights are visible.

- **Conceptual.** Displays objects using smooth shading and the Gooch face style. The Gooch face style transitions between cool and warm colors, rather than dark and light. The effect is less realistic, but it can make the details of the model easier to see.
- **Hidden.** Displays objects using wireframe representation and hides lines representing back faces.
- **Realistic.** Displays objects using smooth shading and materials.
- **Shaded.** Displays objects using smooth shading.
- **Shaded with Edges.** Displays objects using smooth shading and visible edges.
- **Shades of Gray.** Displays objects using smooth shading and monochromatic shades of gray.
- **Sketchy.** Displays objects with a hand-sketched effect by using the Line Extensions and Jitter edge modifiers.
- **Wireframe.** Displays objects using lines and curves to represent the boundaries.
- **X-ray.** Displays objects with partial transparency.



In shaded visual styles, faces are lit 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.

Select a visual style and change its settings at any time. The changes are reflected in the viewports to which the visual style is applied. Any changes you make to the current visual style are saved in the drawing.

For more information about face settings, environment settings, and edge settings, see Customize a Visual Style on page 299.

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 ➤ Visual Styles 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.

To store a visual style on a tool palette

1. Click View tab ➤ Visual Styles 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.

To use a visual style from another drawing

1. Open the drawing that has the visual style that you want to use.
2. Click View tab ➤ Visual Styles 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 in which you want to use the visual style.
8. In the Tool Palette window, right-click the visual style and click Add to Current Drawing.

   The visual style is added to the sample images in the Visual Styles Manager and the ribbon.

To restore the default settings of a predefined visual style

1. Click View tab ➤ Visual Styles panel ➤ ➤ Visual Styles Manager.



2. Right-click a predefined visual style and click Reset to Default.
   (Predefined visual styles display a drawing icon at bottom right in the sample image.)

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.

VSCURRENT

Sets the visual style in the current viewport.

VSSAVE

Saves a visual style.

System Variables

VSLIGHTINGQUALITY

Sets the lighting quality in the current viewport.

Customize a Visual Style

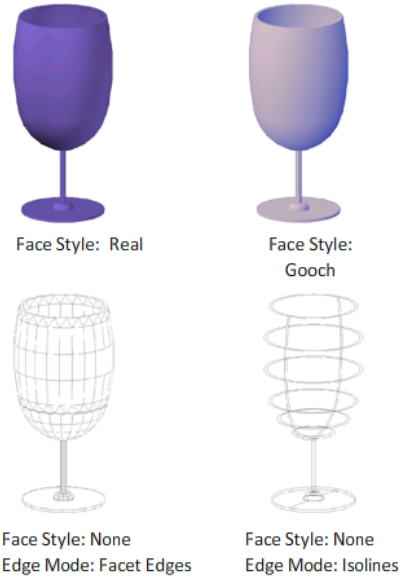
You can create your own visual styles by changing the face and edge settings and using shadows and backgrounds.

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. Realistic (below left) is meant to produce the effect of realism. Gooch (below right) can show details better by softening the contrast between lit areas and shadowed areas. Lit areas use warm tones and shadowed areas use cool tones.



The None face style produces no shading, and displays only edges. Customize edge settings to control whether facet edges or isolines are displayed.

Lighting Quality

Lighting quality determines the smoothness of shaded objects.

Faceted lighting computes a single color for each face. Individual faces appear flat. Smooth lighting smoothes the edges between polygon faces by computing

the colors as a gradient between the faces' vertices. This gives objects a smooth appearance.

For the Smoothest option, the Per-Pixel Lighting setting must be enabled in the Manual Performance Tuning dialog box. The colors are computed for individual pixels, giving a smoother appearance. If not, the Smooth setting is used instead.



Highlights

The size of an object's highlights affect the perception of shininess. A smaller, more intense highlight makes objects appear shinier. The highlight intensity that is set in a visual style does not apply to objects with attached materials.



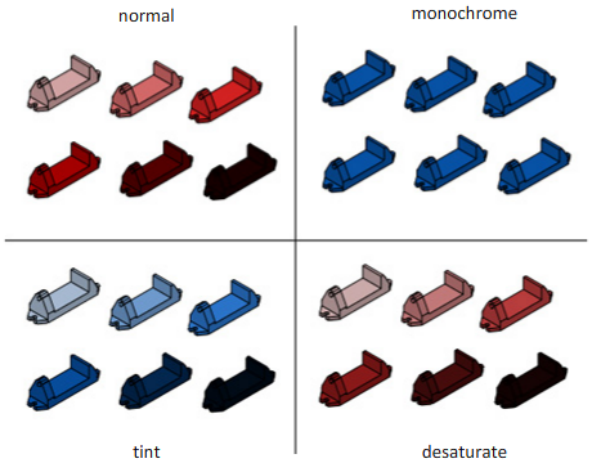
Opacity

The opacity property controls the transparency of objects.



Face Color Modes

Display face colors in the normal way, or specify a face color mode. Monochrome displays faces in the varying shades of a specified color. Tint shades faces by changing the hue and saturation values based on a specified color. Desaturate softens colors.



To change the display of faces from smooth to faceted

1. Click View tab ➤ Visual Styles panel ➤ ➤ Visual Styles Manager.



2. With the current visual style selected in the sample images, under Face Settings, Lighting Quality, change Smooth to Faceted.

To control the face style

1. Click View tab ➤ Visual Styles panel ➤ ➤ Visual Styles Manager.



2. With the current visual style selected in the sample images, under Face Settings, Face Style, select one of the following options:
   - **Realistic.** Shades faces as close as possible to how they 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.

To control the display of colors on faces

1. Click View tab ➤ Visual Styles panel ➤ ➤ Visual Styles Manager.



2. With the current visual style selected in the sample images, under Face Settings, Materials and Color, Face Color Mode, select one of the following options:
   - **Normal**. Does not apply a face color modifier.
   - **Monochrome**. Displays faces in shades of a specified color.
   - **Tint**. Changes the hue and saturation value of face colors based on a specified color.
   - **Desaturate**. Softens the color by reducing its saturation component by 30 percent.
3. If Monochrome or Tint is selected, specify a color. Select Color displays the Select Color dialog box.

To make all faces in a viewport transparent

1. Click View tab ➤ Visual Styles panel ➤ Visual Styles.
2. From the Visual Styles drop-down list, select **X-Ray**.

3 On the Visual Styles panel, set the opacity by dragging the opacity slider.

Quick Reference

Commands

VISUALSTYLES

Creates and modifies visual styles and applies a visual style to a viewport.

VISUALSTYLESCLOSE

Closes the Visual Styles Manager.

VSCURRENT

Sets the visual style in the current viewport.

VSSAVE

Saves a visual style.

System Variables

VSFACECOLORMODE

Controls how the color of faces is calculated.

VSFACEHIGHLIGHT

Controls the display of specular highlights on faces without materials in the current viewport.

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. When Background is set to On in the current visual style, the background is displayed.

To use a background, you first create a named view with a background and set the named view as current in the viewport.

Shadows

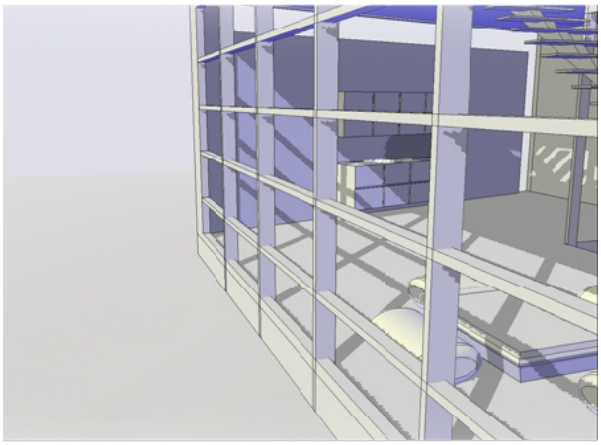
Shaded objects in a viewport can display shadows. Ground shadows are shadows that objects cast on the ground. Mapped object shadows are shadows cast by objects onto other objects. The lighting in the viewport must be from user-created lights or the sun for mapped object shadows to be displayed. Where shadows overlap, they appear darker.

NOTE To display mapped object shadows, hardware acceleration is required. When Enhanced 3D Performance is off, mapped object 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:

- Overview of Lighting on page 2143
- Use Shadows in Rendering on page 2268

To control the display of shadows in a viewport

1. Click View tab ➤ Visual Styles panel ➤ ➤ Visual Styles Manager.



2. With the current visual style selected in the sample images, under Lighting, Shadow Display, select Off, Ground Shadow, or Mapped Object Shadows.

To set a solid as a viewport background

1. Click View tab ➤ Views panel ➤ View Manager.



2. In the View Manager, click the view name you want to change. If the view is not 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.

To set a gradient as a viewport background

1 Click View tab ➤ Views panel ➤ View Manager. 

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.

To set an image as a viewport background

1 Click View tab ➤ Views panel ➤ View Manager. 

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.

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 ➤ View Manager. 

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 middle section, click the Background Override drop-down list and select Sun&Sky.
5. In the Adjust Sun & Sky Background dialog box, specify the desired options for the background.
6. Click OK.
7. In the View Manager, click Set Current. Click OK.

Quick Reference

Commands

VIEW

Saves and restores named model space views, layout views, and preset views.

VISUALSTYLES

Creates and modifies visual styles and applies a visual style to a viewport.

VISUALSTYLESCLOSE

Closes the Visual Styles Manager.

VSCURRENT

Sets the visual style in the current viewport.

VSSAVE

Saves a visual style.

System Variables

CSHADOW

Sets the shadow display property for a 3D object.

LIGHTINGUNITS

Controls whether generic or photometric lights are used, and specifies the lighting units for the drawing.

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.

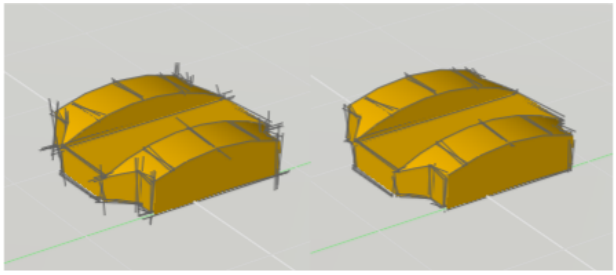
Control the Display of Edges

Different edge types can be displayed using different colors and linetypes. You can also add special effects, such as jitter and line extensions.

In a shaded or wireframe model, the visual style sets the visibility and appearance of isolines, facet edges, silhouette edges, occluded 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 line extension and jitter, produce the appearance of a model that is still in the conceptual phase. Jitter makes lines appear as though they were sketched with a pencil. Line extension produces another kind of hand-drawn effect.

NOTE Plot styles are not available for objects with the Jitter edge modifier applied.

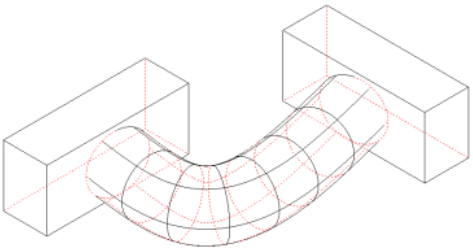


extended and jittered edges jittered edges

Control the display of occluded lines in 2D View

Occluded lines are hidden lines made visible by changing its linetype and color. In 2D View, you can change the display of occluded linetype with

OBSCUREDLTYPE system variable and occluded color with OBSCUREDCOLOR system variable.



To control the display of occluded lines in 2D View, you can:

- Hide them or make them partially visible with dashes and dots.
- Make them completely visible.
- Make them distinctive or indistinctive by changing its color.

NOTE You can only change occluded color when the occluded lines are partially or completely visible.

IMPORTANT After you have changed the settings for occluded lines, use the HIDE command to regenerate the drawing and display the changes.

To control the display of edges and edge colors

1. Click View tab ➤ Visual Styles panel ➤ ➤ Visual Styles Manager.



2. With the current visual style selected in the sample images, in Edge Settings, Show, 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.

To add a line extension effect to edges

1. Click View tab ➤ Visual Styles panel ➤ ➤ Visual Styles Manager.



2. With the current visual style selected in the sample images, in Edge Modifiers, click the Line Extensions Edges button and adjust the amount of line extensions in Line Extensions.

NOTE The line extension is scaled to have an appropriate appearance in viewports of different sizes. The line extension effect is not applied to a line that is less than twice as long as the line extension would be.

To add a jitter effect to edges

1. Click View tab ➤ Visual Styles panel ➤ ➤ Visual Styles Manager.



2. With the current visual style selected in the sample images, in Edge Modifiers, adjust the amount of jitter in Jitter.

To control display of silhouette edges

1. Click View tab ➤ Visual Styles panel ➤ ➤ Visual Styles Manager.



2. With the current visual style selected in the sample images, in Silhouette Edges, Show, select Yes or No.
3. If Yes is selected, specify a width for silhouette edges.

To control the display of occluded edges

1. Click View tab ➤ Visual Styles panel ➤ ➤ Visual Styles Manager.



2 With the current visual style selected to Hidden in sample images, in Edge Settings, Show, click Facet Edges.

3 In Occluded Edges, Show, select Yes.

4 In Color drop-down, select a color from the list.

5 In Linetype drop-down, select a linetype.

To control the display of occluded lines and color

1 Click View tab ➤ Visual Styles panel ➤ ➤ Visual Styles Manager.



2 With the current visual style selected to 2D Wireframe in sample images, in 2D Hide - Occluded Lines, Linetype drop-down, select a linetype.

3 In Color drop-down, select a color from the list.

4 On the command line, enter HIDE and then press Enter for changed occluded line and color settings to take effect.

Quick Reference

Commands

VISUALSTYLES

Creates and modifies visual styles and applies a visual style to a viewport.

VISUALSTYLESCLOSE

Closes the Visual Styles Manager.

VSCURRENT

Sets the visual style in the current viewport.

VSSAVE

Saves a visual style.

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.

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.

VSEDGELEX

Makes edges on 3D objects extend beyond their intersection for a hand-drawn effect.

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.

VSOCCLUDEDCOLOR

Specifies the color of occluded (hidden) lines in the visual style applied to the current viewport.

VSOCCLUDEDEDGES

Controls whether occluded (hidden) edges are displayed.

VSOCCLUDEDLTYPE

Specifies the linetype of occluded (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. On the status bar, click Performance Tuner. 
2. In the Adaptive Degradation and Performance Tuning dialog box, select 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.

To change the adaptive degradation order

1. On the status bar, click 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.
4. Click OK.

Quick Reference

Commands

3DCONFIG

Sets options that affect 3D display performance.

GRAPHICSCONFIG

Sets options for 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

Performance tuning examines your graphics card and 3D display driver and determines 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. Enable these features at your own risk. For information on the options available, see 3DCONFIG.

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 whether effects are enabled. 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
Smooth display: Available and off
Gooch shader: Available and using hardware
Per-pixel lighting: Available and off
Full-shadow display: Available and off
Texture compression: Available and off
Advanced material effects: Available and off
```

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
Smooth display: Not applicable
Gooch shader: Software emulation only
Per-pixel lighting: Not applicable
Full-shadow display: Not applicable
Texture compression: Not applicable
Advanced material effects: Not applicable

Autodesk driver:    Certified
Effect support:
Enhanced 3D Performance: Available
Smooth display: Available
Gooch shader: Available
Per-pixel lighting: Available
Full-shadow display: Available
Texture compression: Available
Advanced material effects: Available
```

NOTE To display mapped object shadows in viewports and to use per-pixel lighting, hardware acceleration is required. When Enhanced 3D Performance is off, mapped object shadows cannot be displayed in viewports. (To access these settings, enter **graphicsconfig** at the Command prompt.)

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 **graphicsconfig** at the Command prompt.)

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. On the status bar, click Performance Tuner. 
2. In the Adaptive Degradation and Performance Tuning dialog box, click View Tune Log.
3. Click OK to exit each dialog box.

To tune performance manually

1. On the status bar, click 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.

To check for graphics card and driver certification updates

1. On the status bar, click 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.

Quick Reference

Commands

3DCONFIG

Sets options that affect 3D display performance.

GRAPHICSCONFIG

Sets options for 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 11

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

Saves and restores named model space views, layout views, and preset views.

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 396. For more information about creating motion path animations, see Create Motion Path Animations on page 400.

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 drop-down ➤ 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.

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 294.

To start a continuous orbit

1. Click View tab ➤ Navigate panel ➤ Orbit drop-down ➤ 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.

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 or 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 396. For more information about creating motion path animations, see Create Motion Path Animations on page 400.

NOTE The Animations panel is hidden by default. To display the Animations panel: In 3D Modeling workspace, click the Render tab, then right-click and choose Show Panels, and then click Animations.

To start walk mode

- Click Render tab ➤ Animations panel ➤ Walk and Fly drop-down ➤ Walk. 

To start fly mode

- Click Render tab ➤ Animations panel ➤ Walk and Fly drop-down ➤ Fly. 

To specify Walk or Fly settings

1. Click Render tab ➤ Animations panel ➤ Walk and Fly drop-down ➤ 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.

To change the viewing position in the Position Locator window

1. Do one of the following:
   - Click Render tab ➤ Animations panel ➤ Walk and Fly drop-down ➤ Walk. 
   - Click Render tab ➤ Animations panel ➤ Walk and Fly drop-down ➤ 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.

To turn off the Position Locator window

1. Click Render tab ➤ Animations panel ➤ Walk and Fly drop-down ➤ Walk and Fly Settings. 
2. In the Walk and Fly Settings dialog box, clear the check mark next to Display Position Locator. Click OK.

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)

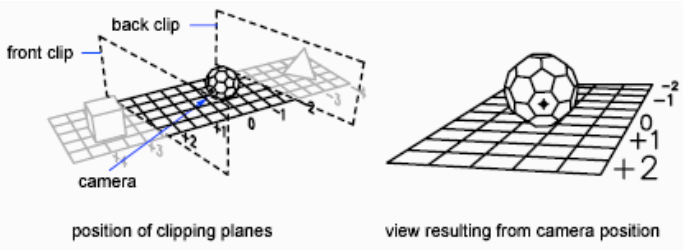
You can change a view without interrupting your current operation using a feature that combines panning and zooming.

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 325.

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 392.

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 392.

To display the default view and UCS

- Click View menu ➤ 3D Views ➤ Plan View ➤ World UCS.

Quick Reference

Commands

DVIEW

Defines parallel projection or perspective views by using a camera and target.

PLAN

Displays an orthographic view of the XY plane 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

Determines whether input to the DVIEW and VPOINT commands is relative to the WCS (default) or the current UCS.

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 tool is a navigation tool that is displayed when you are working in 2D model space or 3D visual style. With ViewCube tool, 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. The ViewCube tool 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 drag or click the ViewCube, 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 pivot point.



To display or hide the ViewCube tool in the current viewport

1. At the command prompt, enter **options** and press Enter.
2. In the Options dialog box ➤ 3D Modeling tab ➤ Display ViewCube or UCS Icon
   - In 2D Model Space, turn the check box on to display the ViewCube tool in all the viewports and drawings in 2D model space
   - In 3D Model Space, turn the check box on to display the ViewCube tool in all the viewports and drawings in 3D model space
3. Click View tab ➤ Windows panel ➤ User Interface drop down ➤ ViewCube.

4 Turn the check box on or off to hide or display the ViewCube in the current viewport

To display or hide the ViewCube tool in all viewports and drawings

1 At the command prompt, enter **options** and press Enter.

2 In the Options dialog box ➤ 3D Modeling tab ➤ Display ViewCube or UCS Icon

- In 2D Model Space, turn the check box on or off to hide and display the ViewCube tool in all the viewports and drawings in 2D model space
- In 3D Model Space, turn the check box on or off to hide and display the ViewCube tool in all the viewports and drawings in 3D model space

To control the position of the ViewCube tool

1 Click View tab ➤ Windows panel ➤ User Interface drop down ➤ 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.

To control the size of the ViewCube tool

1 Click View tab ➤ Windows panel ➤ User Interface drop down ➤ 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.

To control the inactive opacity of the ViewCube tool

1. Click View tab ➤ Windows panel ➤ User Interface drop down ➤ 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.

To display the compass for the ViewCube tool

1. Click View tab ➤ Windows panel ➤ User Interface drop down ➤ 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.

Quick Reference

Commands

NAVVCUBE

Indicates the current viewing direction. Dragging or clicking the ViewCube tool rotates the scene.

OPTIONS

Customizes the program settings.

System Variables

NAVVCUBEDISPLAY

Controls the display of the ViewCube tool in the current visual style and the current viewport.

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. This view is in synchronization with the Go Home view option in the SteeringWheels menu.
- **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.

Quick Reference

Commands

NAVVCUBE

Indicates the current viewing direction. Dragging or clicking the ViewCube tool rotates the scene.

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 parts to click and change the current view of a model. The twenty-six defined parts are categorized into three groups: corner, edge, and face. Of the twenty-six defined parts, six of the parts 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 parts 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.

The outline of the ViewCube tool helps you identify the form of orientation it is in: standard or fixed. When the ViewCube tool is in standard orientation, not orientated to one of the twenty-six predefined parts, 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.



Edge **Corner** **Face**

Drag or Click the ViewCube Tool

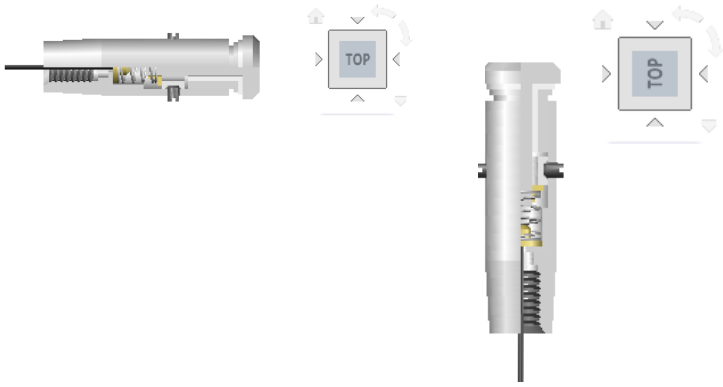
You can also click and drag the ViewCube tool to reorient the view of a model to a custom view other than one of the twenty-six predefined parts. If you drag the ViewCube tool close to one of the preset orientations and it is set to snap to the closest view, the ViewCube tool rotates to the closest preset orientation.

The ViewCube tool reorients the object's view around a pivot point.

- When the object is unselected, drag the ViewCube tool to reorient the object's view around the pivot point displayed at the center of the view.
- When the object is selected, drag the ViewCube tool to reorient the object's view around the pivot point displayed at the center of the selected object.
- When the object is unselected and a pivot point is defined and used by another navigation tool before using the ViewCube tool, drag the ViewCube tool to reorient the object's view around the defined pivot point.

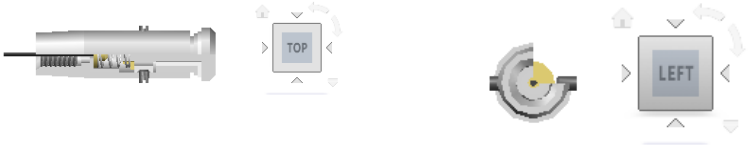
Roll a Face View

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 ➤ Windows panel ➤ User Interface drop down ➤ 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.

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.



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.

To use animated transitions when reorienting a view to a preset orientation

1. Click View tab ➤ Windows panel ➤ User Interface drop down ➤ 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.

To automatically fit the model after a view orientation

1. Click View tab ➤ Windows panel ➤ User Interface drop down ➤ 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.

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.

Quick Reference

Commands

NAVVCUBE

Indicates the current viewing direction. Dragging or clicking the ViewCube tool rotates the scene.

OPTIONS

Customizes the program settings.

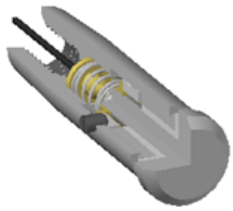
Set the View Projection Mode

View projection produces realistic visual effects of a model.

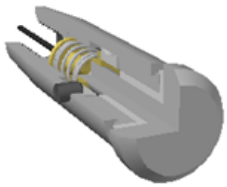
The ViewCube tool supports two view projection modes (Perspective and Orthographic) and a combination of both these modes (Perspective with Ortho faces). Orthographic projection is also referred to as parallel projection. Perspective projected views are calculated based on the distance from a theoretical 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 ➤ Windows panel ➤ User Interface drop down ➤ ViewCube.
2. Right-click the ViewCube tool and click one of the following options:
   - Parallel
   - Perspective
   - Perspective with Ortho Faces

Quick Reference

Commands

NAVVCUBE

Indicates the current viewing direction. Dragging or clicking the ViewCube tool rotates the scene.

OPTIONS

Customizes the program settings.

System Variables

NAVVCUBEDISPLAY

Controls the display of the ViewCube tool in the current visual style and the current viewport.

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 2012 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.

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 Home.

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.

Quick Reference

Commands

NAVVCUBE

Indicates the current viewing direction. Dragging or clicking the ViewCube tool rotates the scene.

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. Select an object or objects and use the ViewCube tool to reorient the model. The model rotates around the center of the view. Calculate the center of the view by the extents of the selected objects.

To examine an individual object with the 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.

Quick Reference

Commands

NAVVCUBE

Indicates the current viewing direction. Dragging or clicking the ViewCube tool rotates the scene.

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 new defined UCS with a name, use the Named option in the UCS command.

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 ➤ Windows panel ➤ User Interface drop down ➤ ViewCube.
2. Right-click 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.

To switch to a different UCS

1. Click View tab ➤ Windows panel ➤ User Interface drop down ➤ 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.

To switch to WCS

1. Click View tab ➤ Windows panel ➤ User Interface drop down ➤ 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.

To set the orientation of the ViewCube tool based on the current UCS or WCS

1. Click View tab ➤ Windows panel ➤ User Interface drop down ➤ 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.

Quick Reference

Commands

NAVVCUBE

Indicates the current viewing direction. Dragging or clicking the ViewCube tool rotates the scene.

OPTIONS

Customizes the program settings.

UCS

Sets the origin and orientation of the current UCS.

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 task specific from which you can navigate and orient a model in different views.

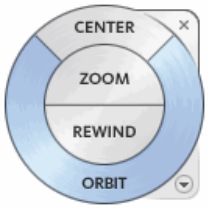
The following illustrations show the different wheels available:



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

NOTE In paper space only 2D Wheel is available.

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

- Click Navigation Bar ➤ Steering Wheels drop-down ➤ Full Navigation.

To close a wheel

Do one of the following:

- Press Esc or Enter
- Click the Close button

To change the size of the wheels

1. Click Navigation Bar ➤ Steering Wheels drop-down . 
2. Click one of the available SteeringWheels.
3. Right-click the wheel and click SteeringWheel Settings.
4. In the SteeringWheels Settings dialog box, under Big Wheels or Mini Wheels, move the Wheel Size slider left or right.

Moving the slider to the left decreases the size of the wheel, while moving the slider to the right increases the size of the wheel.

5 Click OK.

To change the opacity of the wheels

1 Click Navigation Bar ➤ Steering Wheels drop-down ➤ click one of the available SteeringWheels. 

2 Right-click 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.

To enable tooltips for wheels

1 Click Navigation Bar ➤ Steering Wheels drop-down ➤ click one of the available SteeringWheels. 

2 Right-click 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.

To enable messages for wheels

1 Click Navigation Bar ➤ Steering Wheels drop-down ➤ click one of the available SteeringWheels. 

2 Right-click 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.

Quick Reference

Commands

NAVSWHEEL

Provides access to enhanced navigation tools that are quickly accessible from the cursor.

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. This view is in synchronization with the Home view option in the ViewCube tool.
- **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.

Quick Reference

Commands

NAVSWHEEL

Provides access to enhanced navigation tools that are quickly accessible from the cursor.

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 Navigation Bar ➤ SteeringWheels drop-down ➤ 2D Wheel.

Quick Reference

Commands

NAVSWHEEL

Provides access to enhanced navigation tools that are quickly accessible from the cursor.

System Variables

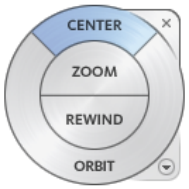
NAVSWHEELMODE

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 at the view's center.

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 Navigation Bar ➤ Steering Wheels drop-down ➤ Basic View Object Wheel . 
- Right-click the wheel, and click Basic Wheels ➤ View Object Wheel.

To switch to the mini View Objects wheel

Do one of the following:

- Click Navigation Bar ➤ Steering Wheels drop-down ➤ Mini View Object Wheel . 
- Right-click the wheel, and click Mini View Object Wheel.

Quick Reference

Commands

NAVSWHEEL

Provides access to enhanced navigation tools that are quickly accessible from the cursor.

System Variables

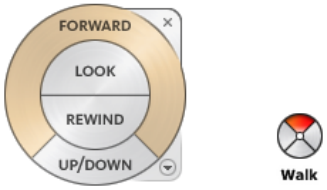
NAVSWHEELMODE

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 Navigation Bar ➤ Steering Wheels drop-down ➤ Basic Tour Building Wheel . 
- Right-click the wheel, and click Basic Wheels ➤ Tour Building Wheel.

To switch to the mini Tour Building wheel

Do one of the following:

- Click Navigation Bar ➤ Steering Wheels drop-down ➤ Mini Tour Building Wheel . 
- Right-click the wheel, and click Mini Tour Building Wheel.

Quick Reference

Commands

NAVSWHEEL

Provides access to enhanced navigation tools that are quickly accessible from the cursor.

System Variables

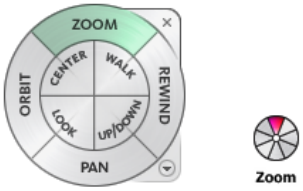
NAVSWHEELMODE

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) contain common 3D navigation tools used for both viewing an object and touring a building. 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 Navigation Bar ➤ Steering Wheels drop-down ➤ Full Navigation Wheel. 
- Right-click the wheel, and click Full Navigation Wheel.

To switch to the mini Full Navigation wheel

Do one of the following:

- Click Navigation Bar ➤ Steering Wheels drop-down ➤ Mini Full Navigation Wheel . 
- Right-click the wheel, and click Mini Full Navigation Wheel.

Quick Reference

Commands

NAVSWHEEL

Provides access to enhanced navigation tools that are quickly accessible from the cursor.

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 and 3Dconnexion 3D mouse. 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.

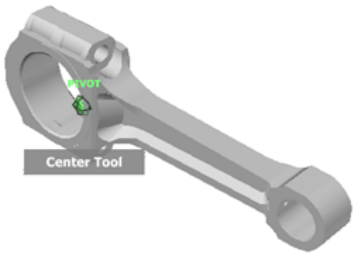
Center Tool

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, drag the cursor over your model. A sphere (pivot point)

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 the cursor is not over the model, the center cannot be set and a prohibited cursor is displayed.



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.

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.

Quick Reference

Commands

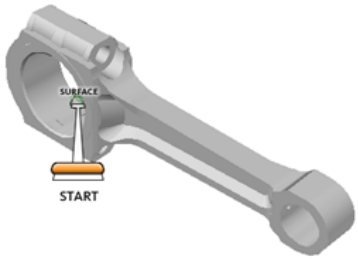
NAVSWHEEL

Provides access to enhanced navigation tools that are quickly accessible from the cursor.

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.

Quick Reference

Commands

NAVSWHEEL

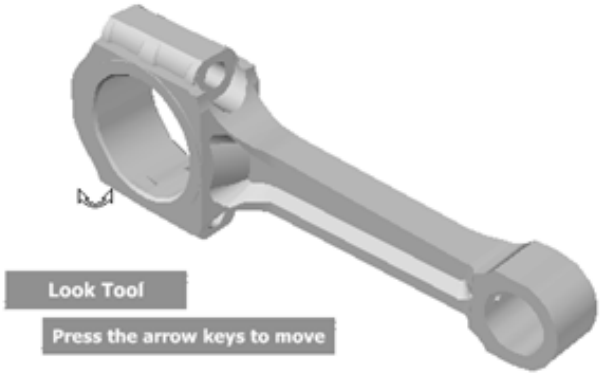
Provides access to enhanced navigation tools that are quickly accessible from the cursor.

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 standing in a fixed location, and looking up, down, 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 the 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.

To look around and walk through the 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.

To invert the vertical axis for the Look tool

1 Click Navigation Bar ➤ Steering Wheels drop-down ➤ click one of the available SteeringWheels. 

2 Right-click 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.

Quick Reference

Commands

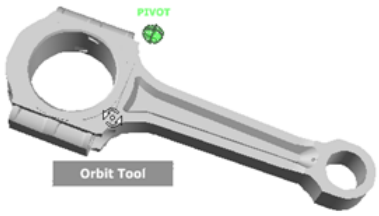
NAVSWHEEL

Provides access to enhanced navigation tools that are quickly accessible from the cursor.

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 press and hold the *CTRL* key at anytime to move the pivot point used by the Orbit tool.This pivot point is used for subsequent navigation until it is moved.

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.

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.

To turn on selection sensitivity for the Orbit tool

1. Click Navigation Bar ➤ Steering Wheels drop-down ➤ click one of the View Object or Full Navigation wheels. 
2. Right-click 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.

To maintain the up direction for the Orbit tool

1 Click Navigation Bar ➤ Steering Wheels drop-down ➤ click one of the View Object or Full Navigation wheels. 

2 Right-click 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.

Quick Reference

Commands

NAVSWHEEL

Provides access to enhanced navigation tools that are quickly accessible from the cursor.

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.

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.

Quick Reference

Commands

NAVSWHEEL

Provides access to enhanced navigation tools that are quickly accessible from the cursor.

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 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.

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.

To specify the display of rewind thumbnails

1. Display a wheel.

2 Right-click 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.

Quick Reference

Commands

NAVSWHEEL

Provides access to enhanced navigation tools that are quickly accessible from the cursor.

System Variables

CAPTURETHUMBNAILS

Specifies if and when thumbnails are captured for the Rewind tool.

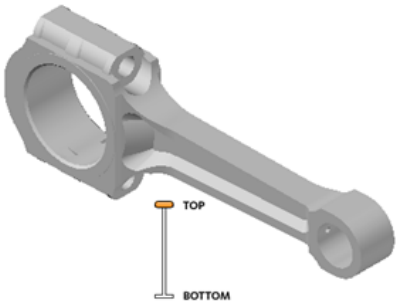
Up/Down Tool

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.

Quick Reference

Commands

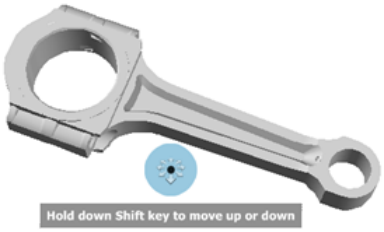
NAVSWHEEL

Provides access to enhanced navigation tools that are quickly accessible from the cursor.

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 base 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.

To change the movement speed for the Walk tool

1 Display a wheel.

2 Right-click 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.

To constrain the walk angle to the ground plane

1 Display a wheel.

2 Right-click 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.

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.

Quick Reference

Commands

NAVSWHEEL

Provides access to enhanced navigation tools that are quickly accessible from the cursor.

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:

- **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 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.

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.

To zoom in and out by scrolling the mouse wheel when a wheel 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.

Quick Reference

Commands

NAVSWHEEL

Provides access to enhanced navigation tools that are quickly accessible from the cursor.

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.

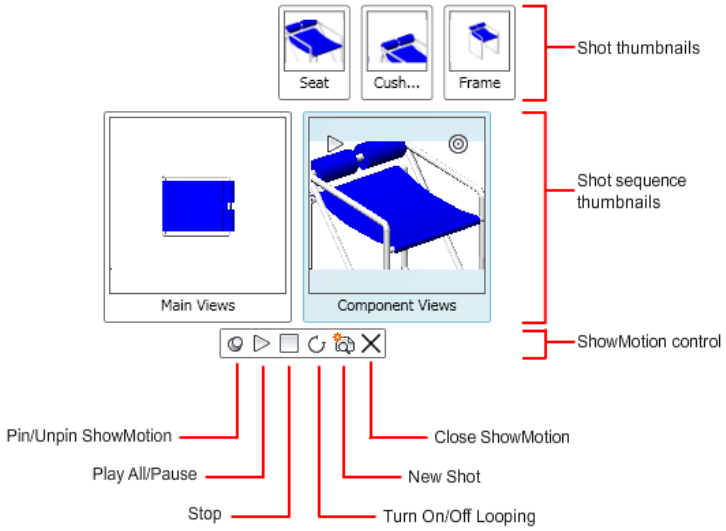
Shot Sequences

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 Navigation Bar ➤ ShowMotion. 

To pin or unpin ShowMotion

- On the ShowMotion control, click Pin ShowMotion or Unpin ShowMotion.

To close ShowMotion

- On the ShowMotion control, click Close ShowMotion or click in the document window when ShowMotion is not pinned.

Quick Reference

Commands

NAVSMOTION

Provides an on-screen display for creating and playing back cinematic camera animations for design review, presentation, and bookmark-style navigation.

System Variables

SHOWMOTIONPIN

Controls the default state of the thumbnail shots.

THUMBSIZE

Specifies the maximum generated size for thumbnail previews in pixels.

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.

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.

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.

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.

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.

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.

To delete a shot

1. On ShowMotion, hover the cursor over the shot you want to delete.
2. Right-click, and click Delete.

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.

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.

Quick Reference

Commands

EDITSHOT

Edits a saved named view with or without motion.

NAVSMOTION

Provides an on-screen display for creating and playing back cinematic camera animations for design review, presentation, and bookmark-style navigation.

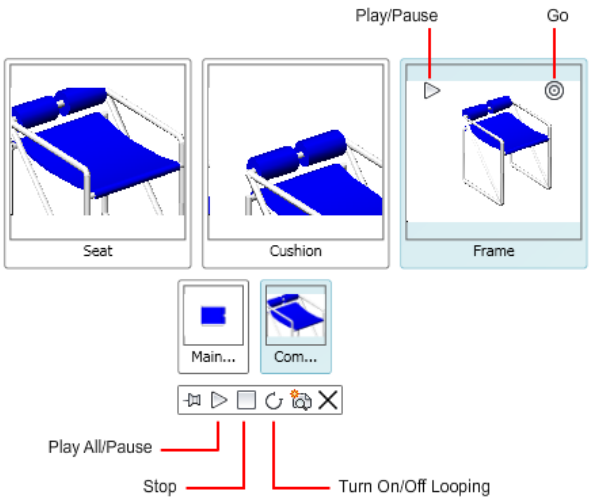
NEWSHOT

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.

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.

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.

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.

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.

Quick Reference

Commands

NAVSMOTION

Provides an on-screen display for creating and playing back cinematic camera animations for design review, presentation, and bookmark-style navigation.

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.

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.

To display a camera

■ Click Render tab ➤ Camera panel ➤ Show 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.

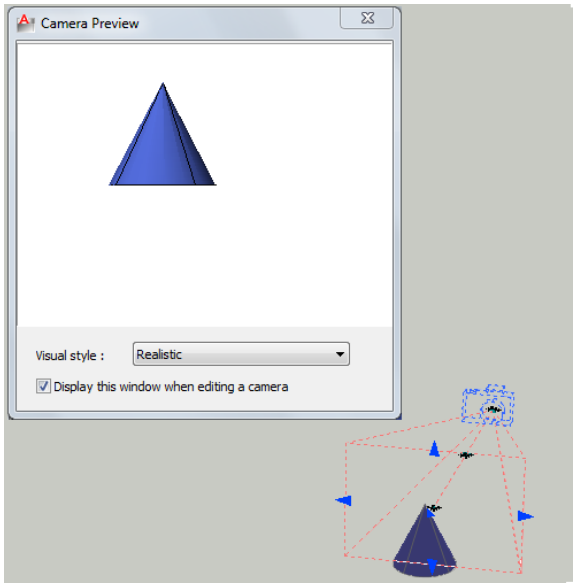
CAMERAHEIGHT

Specifies the default height for new camera objects.

Change Camera Properties

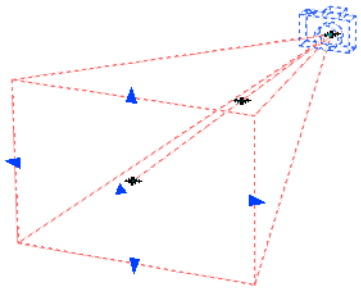
You can modify a camera's lens length, change its front and back clipping planes, name a camera, and turn the display of all cameras on or off in a drawing.

When you select a camera, the Camera Preview dialog box opens to display the camera's view.

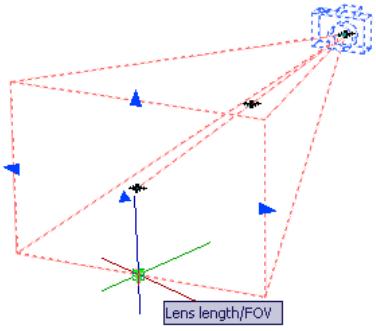


There are several ways to change camera settings:

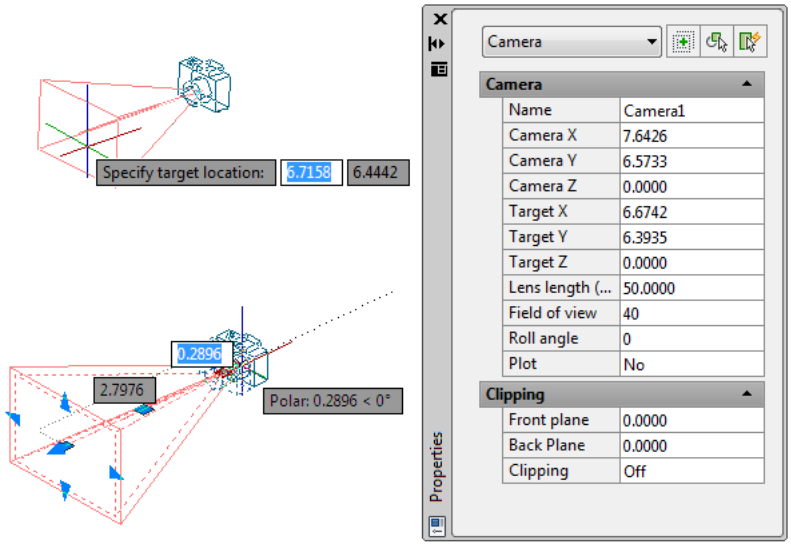
- Click and drag grips to resize or relocate a lens length or field of view.



- Use the Dynamic Input tooltip to enter *X*,*Y*, *Z* coordinate values.



- Modify camera properties in the Camera Properties palette.



NOTE The Camera panel is hidden by default. To display the Camera panel: In 3D Modeling workspace, click the Render tab, then right-click and choose Show Panels, and then click Camera.

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.

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.

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.

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.

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.

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.

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 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 325.

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.

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.

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.

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.

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.

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.

Quick Reference

Commands

ANIPATH

Saves an animation file of a camera moving or panning in a 3D model.

Use Navigation Bar

Unified and product-specific navigation tools can be accessed from the navigation bar.

Available Navigation Tools

The navigation bar is a user interface element where you can access both unified and product-specific navigation tools.

Unified navigation tools (such as Autodesk®, ViewCube ®, ShowMotion®, 3Dconnexion®, and SteeringWheels®) are those that can be found across many Autodesk products. Product-specific navigation tools are unique to a product. The navigation bar floats over and along one of the sides of the current model's window.

You start navigation tools by clicking one of the buttons on the navigation bar or selecting one of the tools from a list that is displayed when you click the smaller portion of a split button.



The following unified navigation tools are available from the navigation bar:

- **ViewCube.** Indicates the current orientation of a model and is used to reorient the current view of a model.
- **SteeringWheels.** Collection of wheels that offer rapid switching between specialized navigation tools.
- **ShowMotion.** User interface element that provides an onscreen display for creating and playing back cinematic camera animations for design review, presentation, and bookmark-style navigation.
- **3Dconnexion.** Set of navigation tools used to reorient the current view of a model with a 3Dconnexion 3D mouse.

The following product-specific navigation tools are available from the navigation bar:

- **Pan.** Moves the view parallel to the screen.
- **Zoom tools.** Set of navigation tools for increasing or decreasing the magnification of the current view of a model.
- **Orbit tools.** Set of navigation tools for rotating the current view of a model.

Use the Navigation Bar in Paper Space

In paper space, only the 2D navigation tools (such as 2D SteeringWheels, Pan, Zoom, and the 2D Mode 3Dconnexion tools) are accessible.



See also:

- Use ViewCube on page 334
- Navigate with SteeringWheels on page 350
- Define and Change Views with ShowMotion on page 380
- Pan or Zoom a View on page 275

- Use 3D Navigation Tools on page 325
- Walk and Fly Through a Drawing on page 328

Quick Reference

Commands

NAVBAR

Provides access to navigation and orientation tools from a single interface.

System Variables

NAVBARDISPLAY

Controls the display of the navigation bar in all viewports.

Reposition and Reorient the Navigation Bar

The position and orientation of the navigation bar can be adjusted by linking it to the ViewCube tool, docking it when the ViewCube tool is not displayed, or freely positioning it along one of the edges of the current window.

When linked to the ViewCube tool, the navigation bar is positioned above or below the ViewCube tool and in a vertical orientation. When not linked or docked, the navigation bar can be freely aligned along one of the edges of the current model's window.

You can specify how the navigation bar can be repositioned from the Customize menu. When the navigation bar is not linked to the ViewCube tool or docked, a grip handle is displayed. Drag the grip handle on the navigation bar to reposition it along one of the sides of the current model's window.



If the side of the window that the navigation bar is aligned to is not long enough to show the entire navigation bar, it is truncated to fit. When truncated, a More Controls button is displayed and replaces the Customize button. When you click the More Controls button, a menu is displayed that contains the navigation tools that are not currently being displayed.

To reposition the navigation bar and ViewCube

1. On the navigation bar, click Customize.
2. Click Customize menu ➤ Docking Positions ➤ check Link to ViewCube.

   When Link to ViewCube is checked, both the navigation bar and ViewCube are repositioned together around the current window. When ViewCube is not displayed, the navigation bar is docked in the same location in which ViewCube would be instead.
3. Click Customize menu ➤ Docking Positions ➤ and then a docking position.

   The navigation bar and ViewCube are repositioned.

To link the position of the navigation bar to ViewCube

1. On the navigation bar, click Customize.
2. Click Customize menu ➤ Docking Positions ➤ check Link to ViewCube.

   When Link to ViewCube is checked, both the navigation bar and ViewCube are repositioned together around the current window.

To freely reposition the navigation bar along the edge of the current window

1. On the navigation bar, click Customize.
2. Click Customize menu ➤ Docking Positions ➤ uncheck Link to ViewCube.

   The grip handle for the navigation bar is displayed along the top of the navigation bar.
3. Click the grip handle and drag the navigation bar along the edge of the window where you want it displayed. Release the button on the pointing device to orient the navigation bar along the edge of the window.
4. Drag the navigation bar along the window's edge to adjust its position along the window's edge.

To display or hide the navigation bar

1. Click View Tab ➤ Windows panel ➤ User Interface drop-down ➤ Navigation Bar.
2. Select or clear Navigation Bar.

Quick Reference

Commands

NAVBAR

Provides access to navigation and orientation tools from a single interface.

System Variables

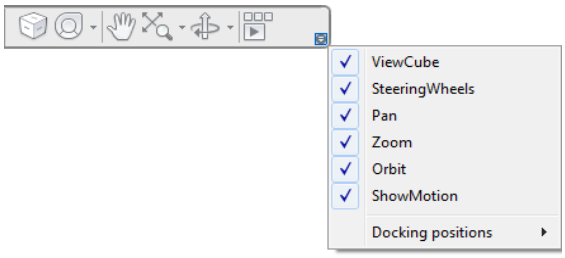
NAVBARDISPLAY

Controls the display of the navigation bar in all viewports.

Control the Display of Navigation Tools on the Navigation Bar

You can control which unified and product-specific navigation tools are displayed on the navigation bar with the Customize menu.

The Customize menu is displayed by clicking the Customize button on the lower-right side of the navigation bar. From the Customize menus, you click the navigation tools that you want displayed on the navigation bar. The position of the navigation tools on the navigation bar is predefined and cannot be changed.



The ViewCube button is displayed on the navigation bar only when the ViewCube tool is hidden in the current window.

The 3Dconnexion button is displayed on the navigation bar only after the 3Dconnexion driver is installed and enabled.

To customize the navigation bar

1. On the navigation bar, click Customize.
2. On the Customize menu, click the navigation tool you want to display on the navigation bar.

A check mark next to a navigation tool's name indicates it is displayed on the navigation bar. Uncheck the navigation tool to remove it from the navigation bar.

Alternately, right click a tool from the navigation bar and click Remove From Navigation Bar.

Quick Reference

Commands

NAVBAR

Provides access to navigation and orientation tools from a single interface.

System Variables

NAVBARDISPLAY

Controls the display of the navigation bar in all viewports.

Use 3Dconnexion 3D mouse to Navigate Views

A 3Dconnexion 3D mouse is used to reorient and navigate a model's view. The device has a pressure sensitive controller cap designed to flex in all directions. Push, pull, twist, or tilt the cap to pan, zoom, and rotate the current view.



When a view change occurs with the 3Dconnexion 3D mouse, the ViewCube tool is reoriented to reflect the current view. You can change the behavior of the 3Dconnexion 3D mouse from the navigation bar.

3Dconnexion options on the navigation bar

Option	Description	Example
Object Mode	Navigates and reorients the view in the direction of the controller cap.	Move the controller cap right to pan the view to the right.
Walk Mode	Simulates walking through a model. The view of the model is moved in the opposite direction of the controller cap. The orientation and height of the current view is maintained.	Move the controller cap up to elevate the view. This makes the model appear to move down.
Fly Mode	Simulates flying through a model. The view of the model is moved in the opposite direction of the controller cap. The orientation and height of the current view is not maintained.	Move the controller cap up to elevate the view. This makes the model appear to move down.
2D Mode	Navigates the view using only 2D navigation options. The view moves in the direction of the controller cap.	Move the controller cap to pan and zoom the view.
3Dconnexion Settings	Displays 3Dconnexion Settings dialog box to change the settings of the 3Dconnexion 3D mouse.	Adjust the navigation speed used for view changes.

In locked viewports, navigation bar options are disabled including 3Dconnexion tool options. Moving the controller cap of the 3Dconnexion 3D mouse will only pan and zoom the object.

NOTE While 3Dconnexion and other navigation tools are in synchronization, using one temporarily disables the other.

Use View Management Keys in 3Dconnexion 3D Mouse

You can access different views (such as Top, Front, Left, Right, or Home) with buttons available on some 3Dconnexion 3D mouse models. Use the Button

Configuration Editor to customize the operations of these buttons. When you click any of these buttons on the device, you can

- **Fit the view to the model extents.** Rotates the view of the object around the center of the scene and zooms out to fit the scene into the viewport.
- **Reorient the current view to a preset view.** Returns the view of the object to a predetermined view.
- **Maintain selection sensitivity.** Reorients the model around a defined pivot point based on the current selection.

Quick Reference

Commands

NAVBAR

Provides access to navigation and orientation tools from a single interface.

System Variables

NAVBARDISPLAY

Controls the display of the navigation bar in all viewports.

Display Multiple Views in Model Space

12

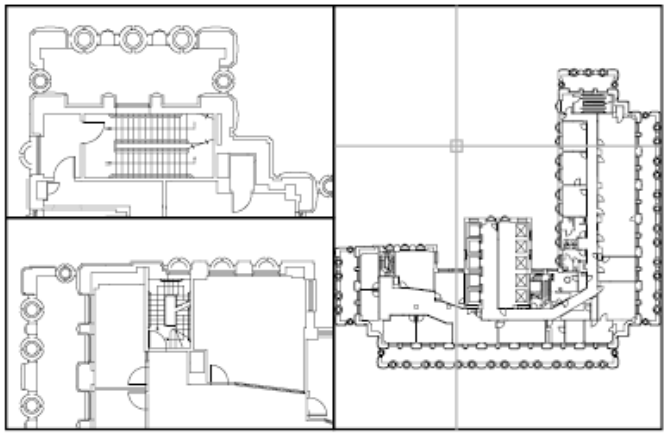
To see several views at the same time, you can split the drawing area of the Model layout into separate viewing areas called *model space viewports*. You can save arrangements of model space viewports for reuse at any time.

Set Model Space Viewports

On the Model layout, 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 layout, 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 layout 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 named (paper space) layout. 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

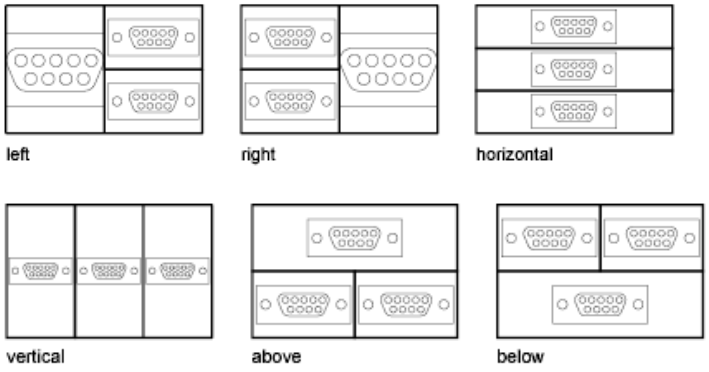
With model space viewports, you can do the following:

- 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 layout or insert it on a named layout.

Setting up different coordinate systems in individual viewports is useful if you typically work on 3D models. See Assign UCS Definitions to Viewports on page 599.

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 layout

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.
   - Click View menu ➤ Viewports ➤ 3 Viewports.
   - Click Viewmenu ➤ Viewports ➤ 4 Viewports.
3. At the Next prompt, specify the arrangement of the new viewports.

To join two viewports on the Model layout

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.

To restore a single viewport on the Model layout

- Click View tab ➤ Viewports panel ➤ Viewports ➤ Viewport Configurations List ➤ Single

To switch from a named layout to the Model layout

- Click the Model tab at the bottom of the drawing area.

Quick Reference

Commands

MODEL

Switches from a named (paper space) layout to the Model layout.

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 layout in the drawing.

TILEMODE

Makes the Model tab or the last layout tab current.

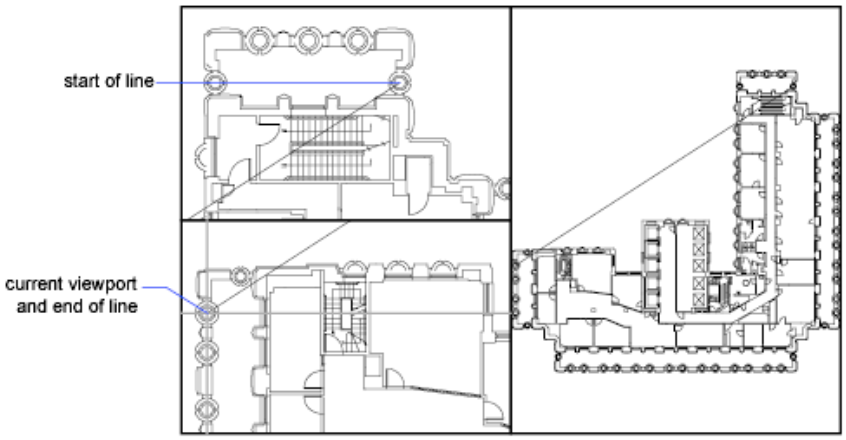
Select and Use the Current Viewport

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

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.

Save and Restore Model Layout Viewport Arrangements

Arrangements of model viewports can be saved and restored by name.

You do not 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 layout can be inserted on a named layout.

To save and name a viewport arrangement

1. Click View tab ➤ Viewports panel ➤ Named. 
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 layiout.

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.

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.

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.

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.

Part 5: Organize Drawings and Layouts

Create Single-View Drawings (Model Space)

13

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 Model and named layouts.

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. 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 annotative on page 2289 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 you output the drawing.

For information about using annotative objects and scaling annotations automatically, see Scale Annotations on page 1324.

See also:

- Create Multiple-View Drawing Layouts (Paper Space) on page 431
- Work with Sheets in a Sheet Set on page 471

Quick Reference

Commands

MODEL

Switches from a named (paper space) layout to the Model layout.

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.

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 annotative on page 2289 objects if you want to scale annotations automatically. For information about using annotative objects and scaling annotations automatically, see Scale Annotations on page 1324.

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 annotative on page 2289 objects if you want to scale annotations automatically. For information about using annotative objects and scaling annotations automatically, see Scale Annotations on page 1324.

Determine the Scale Factor for Plotting

To plot your drawing from the Model layout, 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 207

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.

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.

LTSCALE

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

Sets the hatch pattern scale factor.

HPSPACE

Sets the hatch pattern line spacing for user-defined patterns.

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.

14 Create Multiple-View Drawing Layouts (Paper Space)

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

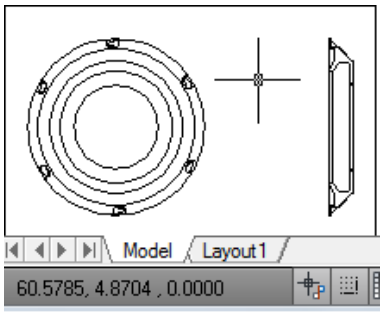
There are two distinct working environments, or "spaces," in which you can create objects in a drawing.

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 named layout tabs.

NOTE Layouts can be represented as tabs which can be hidden, appearing instead as buttons on the status bar at the bottom-center of the application window.

Working in model space, you draw a model of your subject at 1:1 scale. Working on a named layout, you 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

LAYOUT

Creates and modifies drawing layouts.

LAYOUTWIZARD

Creates a new layout tab and specifies page and plot settings.

MODEL

Switches from a named (paper space) layout to the Model layout.

MSPACE

In a layout, switches from paper space to model space in a layout 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

In a layout, switches from model space in a viewport to paper space.

VPORTS

Creates multiple viewports in model space or paper space.

VPLAYER

Sets layer visibility within viewports.

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.

Understand the Layout Process

When you use a named layout to prepare your drawing for output, you follow a series of steps in a process.

You design the subject of your drawing in model space and prepare it for output on a named layout in paper space. A drawing always has at least one named layout.

Before you can use a layout, it must be initialized. A layout does not contain any page setup information before it is initialized. Once initialized, layouts can be drawn upon and output.

A layout can only be added to a sheet sets as sheets after it has been initialized and saved.

Process Summary

When you prepare a layout, you typically step through the following process:

- Create a model of your subject in model space.

- Initialize a named layout.
- Specify layout page settings such as output device, paper size, drawing area, output 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.
- Output your layout.

You can also use annotative on page 2289 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 1324.

The other topics in this chapter provide additional detail on how to create, use, and modify layouts and layout viewports.

Quick Reference

LAYOUT

Creates and modifies drawing layouts.

LAYOUTWIZARD

Creates a new layout tab and specifies page and plot settings.

MODEL

Switches from a named (paper space) layout to the Model layout.

MSPACE

In a layout, switches from paper space to model space in a layout 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

In a layout, switches from model space in a 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.

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.

Work with Model Space and Paper Space

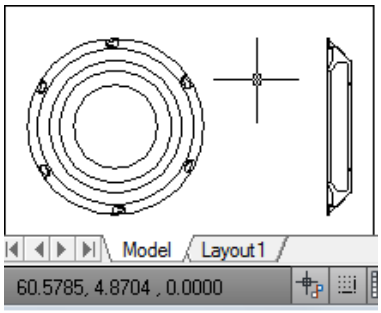
There are several benefits to switching between model space and paper space to perform certain tasks. Use model space for creating and editing your model. Use paper space for composing your drawing sheet and defining views.

Work in Model Space

By default, you start working in a limitless drawing area called *model space*. In model space, you draw, view, and edit your model.

You first decide whether one unit represents one millimeter, one centimeter, one inch, one foot, or whatever unit is most convenient or customary in your business. You then create your model at 1:1 scale.

In model space, 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

MODEL

Switches from a named (paper space) layout to the Model layout.

MSPACE

In a layout, switches from paper space to model space in a layout viewport.

PSPACE

In a layout, switches from model space in a viewport to paper space.

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

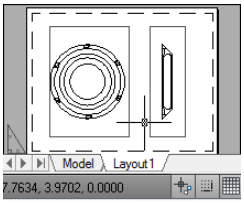
Makes the Model tab or the last layout tab current.

Work on a Named Layout

Named layouts 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 actual distance on a sheet of paper. The units will be in either millimeters or inches, depending on how you configure your page setup.

On a named layout, 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.



Create Additional Named Layouts

By default, a new drawing starts with two named layouts, named Layout1 and Layout2. If you use a drawing template or open an existing drawing, the layouts in your drawing may be named differently.

You can create a new layout using one of the following methods:

- Add a new layout 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 and its settings from the current drawing file.
- Import a layout 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 page setup settings. However, to avoid confusion when transmitting and publishing drawings, it is usually recommended that you create only one named 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 move an object from model space to paper space (or vice versa)

1. Click Home tab ➤ Modify panel ➤ Change Space.
2. Select one or more objects to move.
3. Press Enter to end the command.

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.

Quick Reference

LAYOUT

Creates and modifies drawing layouts.

LAYOUTWIZARD

Creates a new layout tab and specifies page and plot settings.

MODEL

Switches from a named (paper space) layout to the Model layout.

MSPACE

In a layout, switches from paper space to model space in a layout 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

In a layout, switches from model space in a viewport to paper space.

VPORTS

Creates multiple viewports in model space or paper space.

VPLAYER

Sets layer visibility within viewports.

CTAB

Returns the name of the current layout in the drawing.

CVPORT

Displays the identification number of the current viewport.

LAYOUTCREATEVIEWPORT

Controls whether a viewport is created automatically on each new layout added to a drawing.

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.

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 Maximize Viewport button on the status bar to make the layout viewport fill the application window. The center point and the layer visibility settings of the layout 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.

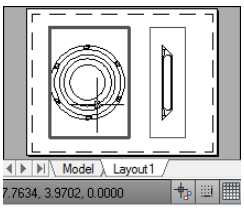
NOTE If you use PLOT while a viewport is maximized, the layout is restored before the Plot dialog box is displayed. If you save and close the drawing while a viewport is maximized, the drawing opens with the named layout restored.

If you choose to switch to the default model space 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.

Quick Reference

MODEL

Switches from a named (paper space) layout to the Model layout.

MSPACE

In a layout, switches from paper space to model space in a layout viewport.

PSPACE

In a layout, switches from model space in a viewport to paper space.

VPMAX

Expands the current layout viewport for editing.

VPMIN

Restores the current layout viewport.

VPMAXIMIZEDSTATE

Indicates whether the viewport is maximized or not.

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.

Object Type	Representation in Exported Drawing
Constraints	Dimensional constraints are removed.
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. **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.

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**.

Quick Reference

EXPORTLAYOUT

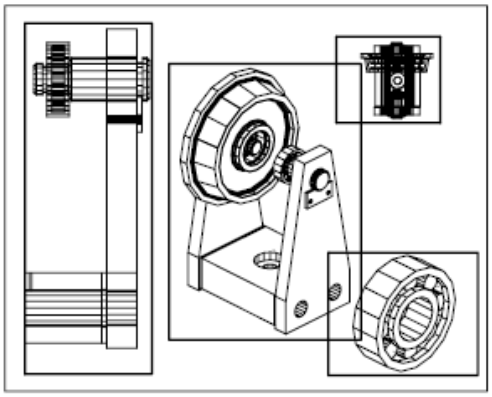
Creates a visual representation of the current layout in the model space of a new drawing.

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 output your drawing, you can turn off the layer and output the layout without 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. 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.

To create a viewport configuration on a layout

1. Click a layout tab.
2. 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.

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.

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.

To clip a layout viewport boundary

1 Click View tab ➤ Viewports panel ➤ 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.

Quick Reference

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.

MAXACTVP

Sets the maximum number of viewports that can be active at one time in a layout.

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 output 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 Scale on the status bar

NOTE You can modify the list of scales that are displayed in all view and plot scale lists with SCALELISTEDIT. After you add a new scale to the default scale list, you can use the Reset button in the Edit Drawing Scales dialog box to add the new scale to your drawing.

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.

When creating a new drawing based on a template, the scales in the template are used in the new drawing. The scales in the user profile are not imported.

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.

Annotative Objects and Scaling

Annotative objects are defined at a paper height instead of a model size and assigned one or more scales. These objects are scaled based on the current annotation scale setting and automatically displayed at the correct size in the layout or when plotted. The annotation scale controls the size of the annotative objects relative to the model geometry in the drawing.

You can specify the default list of scales available for layout viewports, page layouts, and plotting in Default Scale List dialog box.

For more information about annotation scaling, see Scale Annotations on page 1324.

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.

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.

Quick Reference

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.

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.

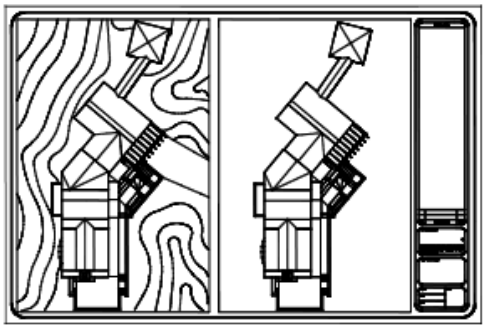
See also:

- Display Annotative Objects on page 1351

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. 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.

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.

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.

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.

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.

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.

Quick Reference

LAYER

Manages layers and layer properties.

VPLAYER

Sets layer visibility within viewports.

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 output to 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 1681

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

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

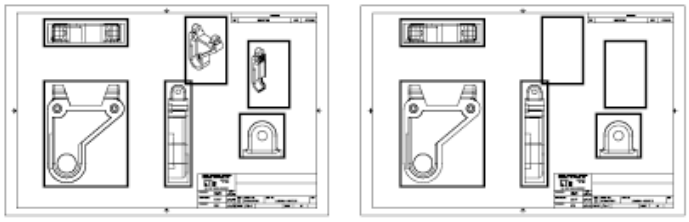
PROPERTIES

Controls properties of existing objects.

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.

Quick Reference

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.

MAXACTVP

Sets the maximum number of viewports that can be active at one time in a layout.

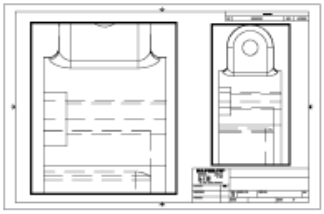
Scale Linetypes in Layout Viewports

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.

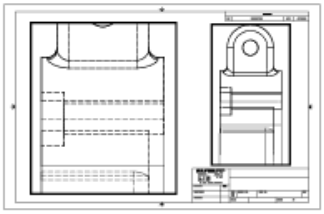
You can set thePSLTSCALE system variable to maintain the same linetype scaling for objects displayed at different zoom factors in a layout and in a layout viewport. For example, with PSLTSCALE set to 1 (default), set the current linetype to dashed, and then draw a line in a paper space layout.

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 1648

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.

Quick Reference

LINETYPE

Loads, sets, and modifies linetypes.

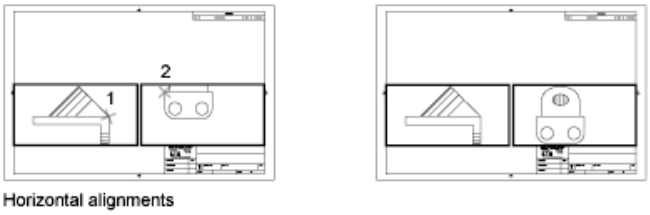
PSLTSCALE

Controls the linetype scaling of objects displayed in paper space viewports.

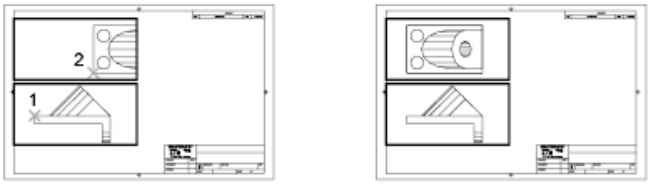
Align Views in Layout Viewports

You can arrange the elements of your drawing by aligning the view in one layout viewport with the view in another viewport.

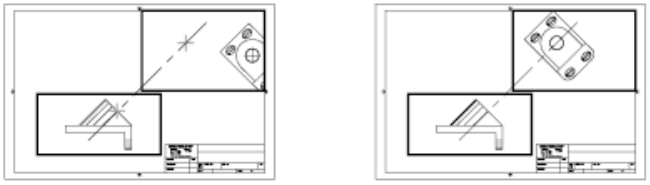
For angled, horizontal, and vertical alignments, you can move each layout viewport relative to distances defined by the model-space geometry displayed.



Horizontal alignments



Vertical alignments



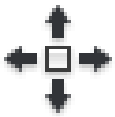
Angled alignments

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 drafting aids 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.

8 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

MOVE

Moves objects a specified distance in a specified direction.

MVSETUP

Sets up the specifications of a drawing.

UCS

Sets the origin and orientation of the current UCS.

UCSICON

Controls the visibility, placement, appearance, and selectability of the UCS icon.

UCSMAN

Manages UCS definitions.

UCSICON

Controls the visibility, placement, appearance, and selectability of the UCS icon.

UCSVP

Determines whether the UCS in viewports remains fixed or changes to reflect the UCS of the current viewport.

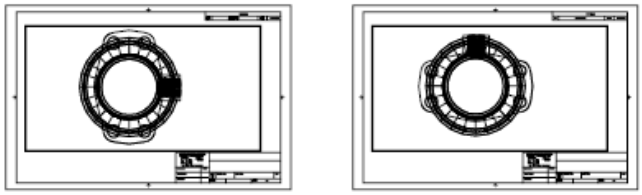
Rotate Views in Layout Viewports

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.



Another way is to use 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 ➤ Z.

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.

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

MVSETUP

Sets up the specifications of a drawing.

PLAN

Displays an orthographic view of the XY plane of a specified user coordinate system.

UCS

Sets the origin and orientation of the current UCS.

UCSICON

Controls the visibility, placement, appearance, and selectability of the UCS icon.

UCSMAN

Manages UCS definitions.

UCSICON

Controls the visibility, placement, appearance, and selectability 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.

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 Files tab (Options

dialog box). 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.

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.

Quick Reference

ADCENTER

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

LAYOUT

Creates and modifies drawing layouts.

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 or current named (paper space) layout.

PURGE

Removes unused items, such as block definitions and layers, from the drawing.

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.

Work with Sheets in a Sheet Set

15

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.

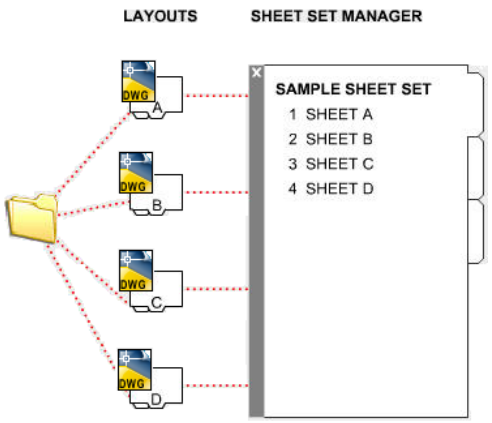
Sheet sets facilitate the organization and management of drawings in a project, and improve the communication in a work group.

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

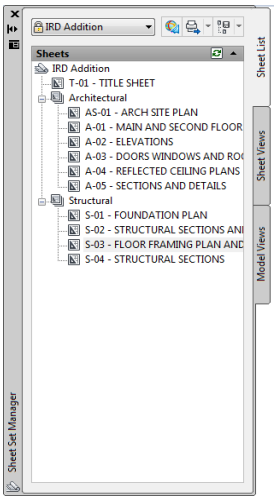
Controls how the status data in a sheet set is refreshed.

SSMSTATE

Indicates whether the Sheet Set Manager window is open or closed.

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.

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.

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.

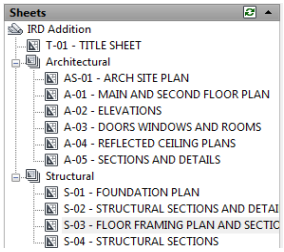
SSMSTATE

Indicates whether the Sheet Set Manager window is open or closed.

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

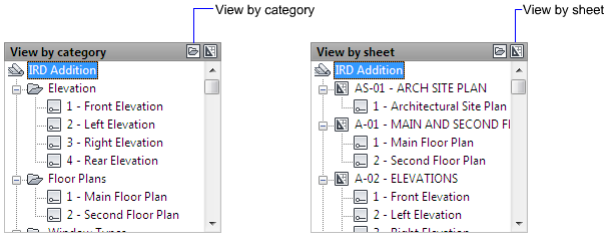
Sheet subsets are often associated with a discipline such as architecture or mechanical design. For example, in architecture, you might use a subset named Structural; and in mechanical design, you might use a subset called Standard Fasteners. In some cases, you might also find it useful to create subsets 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 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.

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.

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.

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.

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.

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.

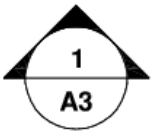
NOTE After creating a named model space view, you must save the drawing to add the view to the Model Views tab. Click Refresh on the Model Views tab to update the Sheet Set Manager tree view.

- **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 1458.

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
- Use Fields in Text on page 1458

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.

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.

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.

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.

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.

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.
5. Click OK.

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.

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.

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.

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.

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.

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.

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.

Quick Reference

Commands

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Opens a selected sheet set.

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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.

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 1742. 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 1716.

  For information on DWFx files, see Plot DWFx Files on page 1717.

- **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 on page 2109.

- **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 1655.

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.

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.

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.

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.

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.

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.

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.

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.

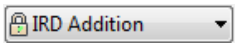
Use Sheet Sets in a Team

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.

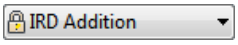
Any changes that any team member makes opens the DST file briefly and updates the information stored in that file. When the DST file is opened, a lock icon is displayed next to the sheet set name in the top left corner of the Sheet Set Manager.



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 an older version of AutoCAD LT or 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.

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.

Part 6: Create and Modify Objects

Control the Properties of Objects

16

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, transparency, and plot style.

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, transparency, 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.

Object properties can be specified and changed with the Properties and Quick Properties palettes, and with the controls available on the Home tab, Properties panel of the ribbon.

See also:

- Control the Color and Linetype Properties in Blocks on page 934

Quick Reference

CHPROP

Changes the properties of an object.

PROPERTIES

Controls properties of existing objects.

PROPERTIESCLOSE

Closes the Properties palette.

QUICKPROPERTIES

Displays quick property data for selected objects.

CECOLOR

Sets the color of new objects.

CELTYPE

Sets the linetype of new objects.

CELWEIGHT

Sets the lineweight of new objects.

CETRANSPARENCY

Sets the transparency level for new objects.

CLAYER

Sets the current layer.

CPLOTSTYLE

Controls the current plot style for new objects.

PLOTTRANSPARENCYOVERRIDE

Controls whether object transparency is plotted.

PROPOBJLIMIT

Limits the number of objects that can be changed at one time with the Properties and Quick Properties palettes.

QPMODE

Controls whether the Quick Properties palette is displayed when objects are selected.

RIBBONCONTEXTSELLIM

Limits the number of objects that can be changed at one time with the ribbon property controls or a contextual tab.

TRANSPARENCYDISPLAY

Controls whether the object transparency is displayed.

Display and Change the Properties of Objects

You can display and change the current properties for any object in your drawing.

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.

For information about controlling the display and behavior of the Properties palette, see Specify the Behavior of Dockable Windows on page 175.

Control Double-Click Behavior

You can double-click most objects to open the Quick Properties palette when the DBLCLKEDIT and PICKFIRST system variables are turned on (the default).

Several types of objects open an editor or start an object-specific command when you double-click them instead of the Quick Properties palette. These types of objects include blocks, polylines, splines, text, and more.

Use the Customize User Interface dialog box (CUI) to control what palette or command is used when double-clicking each type of object.

Use the Properties Panel on the Ribbon

On the Home tab, the Properties panel provides a convenient way to verify or change the settings for properties such as color, layer, and linetype. The Properties panel operates similarly to the Properties palette.



- If *no* objects are selected, the panel displays the default properties for objects created in the future.
- If one or more objects are selected, the controls display the current properties for the selected objects.
- If one or more objects are selected and their properties vary, the controls for those properties will be blank.
- If one or more objects are selected, and a property is changed on the ribbon, the selected objects will change to the specified value.

Maintaining Performance When Changing Object Properties

For performance reasons, the number of objects that can be changed at one time by the Properties panel and the Layer control on the ribbon, the Properties palette, and the Quick Properties palette are limited. The default values of the system variables that control these limits are

- Properties panel: 2,500 objects (RIBBONCONTEXTSELLIM)
- Layer control: 2,500 objects (RIBBONCONTEXTSELLIM)
- Properties palette: 25,000 objects (PROPOBJLIMIT)
- Quick Properties palette: 25,000 objects (PROPOBJLIMIT)

If you need to change the properties of a larger number of objects, you can either change the default values for their system variables, or you can use the CHANGE or CHPROP commands.

See also:

- Control the Color and Linetype Properties in Blocks on page 934
- Select Objects by Properties on page 749
- Set Interface Options on page 149
- Quick Properties in the *Customization Guide*

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.

Quick Reference

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.

QUICKPROPERTIES

Displays quick property data for selected objects.

SETBYLAYER

Changes the property overrides of selected objects to ByLayer.

CETRANSPARENCY

Sets the transparency level for new objects.

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.

PROPOBJLIMIT

Limits the number of objects that can be changed at one time with the Properties and Quick Properties palettes.

QPLOCATION

Sets the location for the Quick Properties palette.

QPMODE

Controls whether the Quick Properties palette is displayed when objects are selected.

RIBBONCONTEXTSELLIM

Limits the number of objects that can be changed at one time with the ribbon property controls or a contextual tab.

SETBYLAYERMODE

Controls which properties are selected for the SETBYLAYER command.

TRANSPARENCYDISPLAY

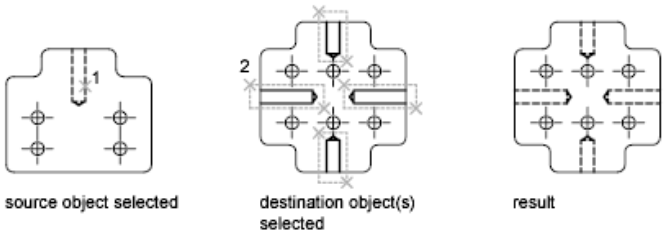
Controls whether the object transparency is displayed.

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, transparency, 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 ➤ Clipboard 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.

Quick Reference

MATCHPROP

Applies the properties of a selected object to other objects.

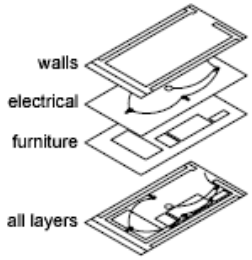
Work with Layers

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. 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

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.

For information about controlling the display and behavior of the Layer Properties manager, see Specify the Behavior of Dockable Windows on page 175.

Quick Reference

CLASSICLAYER

Opens the legacy Layer Properties Manager.

LAYER

Manages layers and layer properties.

LAYERPALETTE

Opens the modeless Layer Properties Manager.

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.

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.

Control Transparency on Layers

Set the transparency of layers and layout viewports to enhance drawings by reducing the visibility of all object on specific layers as needed. Set layer (or layout viewport) transparency in the Layer Properties Manager.

After you apply transparency to a layer, all objects added to that layer are created at the same level of trasparency. The transparency property for all objects on the layer is set to ByLayer.

Assign a Default Color and Linetype to a Layer

Each layer has associated properties such as color, linetype, and transparency that are assumed by all objects on that layer when the setting is ByLayer.

For example, if the Properties palette set to BYLAYER when no object is selected, 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 to the Properties palette when no objects are selected, that color is used for all new objects, overriding the default color for the current layer. The same is true for Linetype, Lineweight, Transparency, and Plot Style properties on the Properties palette.

The BYBLOCK setting should be used only for creating blocks. See Control the Color and Linetype Properties in Blocks on page 934.

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, transparency, 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 529 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.

When you lock a layer that contains transparent objects, the visibiltiy of those objects is further reduced by the specified locked layer fading value.

NOTE Grips are not displayed on objects that are on locked layers.

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.

To control the transparency of objects or layout viewports on a layer

1. Click Home tab ➤ Layers panel ➤ Layer Properties. 
2. In the Layer Properties Manager, select a layer. Click the Transparency or the VP Transparency column.

   NOTE The VP Transparency column is only available when a layout tab is active.

3. In the Layer Transparency dialog box, click the drop-down list to select a transparency level, or enter a value between 0 and 90 .
4. Click OK.

TIP To set the same trasparency level for multiple selected layers, press CTRL and click to select more than one layer. Then click the Transparency or the VP Transparency column to display the Layer Transparency dialog box.

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.

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.

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.

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 OK.

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 OK.

Quick Reference

CLASSICLAYER

Opens the legacy 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.

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.

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, this makes it easier to locate and manipulate groups of layers at a time.

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 Drawing Template File on page 211.

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 OK.

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 OK.

To purge all unused layers

1 Click File ➤ Drawing Utilities ➤ 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.

Quick Reference

CLASSICLAYER

Opens the legacy 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.

CLAYER

Sets the current layer.

LAYERDLGMODE

Sets the flavor of the Layer Properties Manager that is defined for the use of the LAYER command.

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 change layer properties in the Layer Properties Manager. Click the icons to change settings, and rename layers or change descriptions.

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 layer change or set of changes made. 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.

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 OK.

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 OK.

To rename more than one layer

1. Click Format ➤ Rename. 
2. In the Rename dialog box, in the Named Objects list, 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 OK.

Quick Reference

CHANGE

Changes the properties of existing objects.

CHPROP

Changes the properties of an object.

CLASSICLAYER

Opens the legacy 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 on and off the tracking of changes made to layer settings.

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.

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.

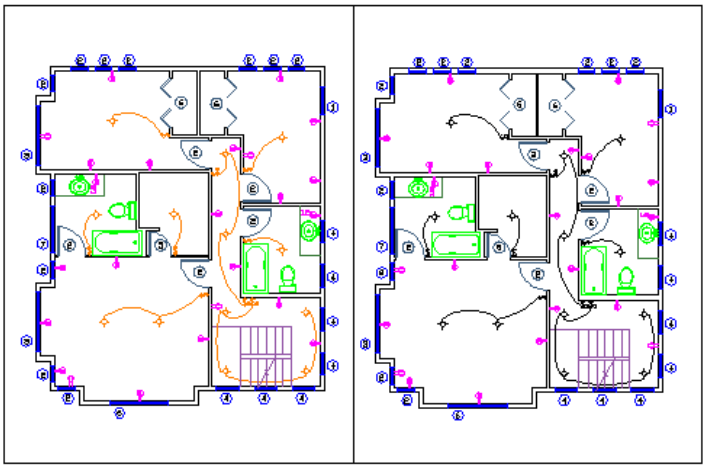
SETBYLAYERMODE

Controls which properties are selected for the SETBYLAYER command.

Override Layer Properties in Viewports

You can display objects differently by setting property overrides for color, linetype, lineweight, transparency, 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.

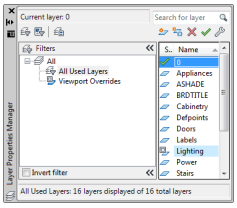


Property override settings for color and lineweight were set on the Wiring layer for the viewport on the left. Notice the wiring is a different color and lineweight than in the right viewport.

When the Layer Properties Manager is accessed from a layout tab, five columns for layer property overrides are displayed

- VP Color
- VP Linetype
- VP Lineweight
- VP Transparency
- 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

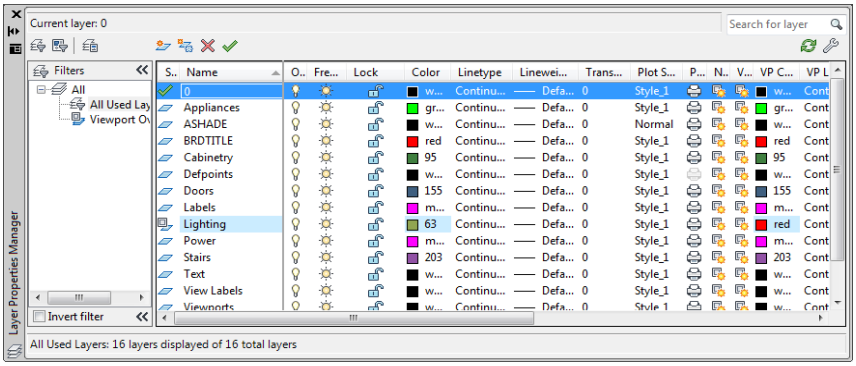
Layers containing property overrides are identifiable in the Layer Properties Manager when accessed from a layout tab. You can see which layers have overrides by the following:

- 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.

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.

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.

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.

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.

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.

To control the transparency of objects or layout viewports on a layer

1. Click Home tab ➤ Layers panel ➤ Layer Properties. 
2. In the Layer Properties Manager, enter a value between 0 and 90 in the Transparency or VP Transparency columns for each layer you want to make transparent.

   NOTE The VP Transparency column is only available when a Layout tab is active.

3. Press Enter.

To not display or plot layer viewport overrides

1. At the Command prompt, enter vplayeroverridesmode.
2. Enter **0**.

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.

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.

Quick Reference

CHPROP

Changes the properties of an object.

CLASSICLAYER

Opens the legacy 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 on and off the tracking of changes made to layer settings.

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.

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.

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 542 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
* (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

Character	Definition
~ (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 542

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.

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 OK.

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 OK.

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 OK.

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.

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.

Quick Reference

CLASSICLAYER

Opens the legacy Layer Properties Manager.

LAYER

Manages layers and layer properties.

LAYERPALETTE

Opens the modeless Layer Properties Manager.

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.

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.

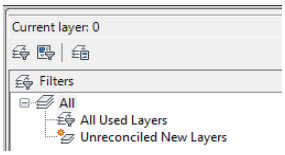
Unreconciled layers are new layers that have been added to the drawing or to attached xrefs since the layer list was last evaluated. The layer list is checked for new layers when a command, such as PLOT is used. In new drawings, the layer baseline is created when the drawing is saved or plotted for the 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

CLASSICLAYER

Opens the legacy Layer Properties Manager.

LAYER

Manages layers and layer properties.

LAYERPALETTE

Opens the modeless Layer Properties Manager.

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.

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.

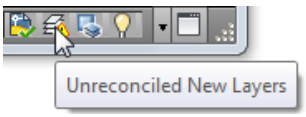
You can control when to evaluate a drawing for new layers. You can specify which commands, such as SAVE or PLOT, trigger the program to check the layer list and alert you of new layers. This can include new layers that have been added to attached xrefs.

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.

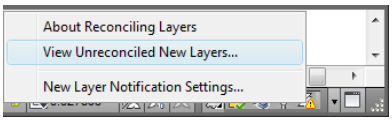
When layer notification is turned on, an Unreconciled New Layers icon displays on the status bar.



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.

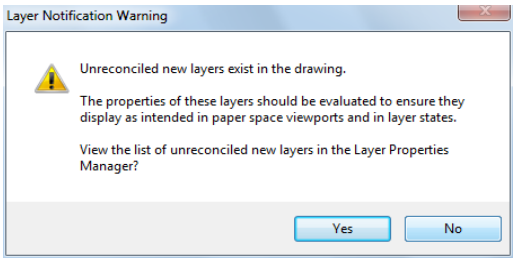
The new layers are *unreconciled* because they have not yet been reviewed. The process of manually marking them as *reconciled* removes them from the Unreconciled New Layers list. (See Reconcile New Layers on page 542 for more information.) Until you reconcile the layers, the notification bubble will 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.

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.

Quick Reference

CLASSICLAYER

Opens the legacy Layer Properties Manager.

LAYER

Manages layers and layer properties.

LAYERPALETTE

Opens the modeless Layer Properties Manager.

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.

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, transparency, 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.

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.

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.

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.

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.

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.

Quick Reference

LAYERSTATE

Saves, restores, and manages named layer states.

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.

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.

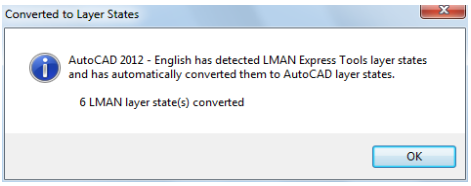
When importing a layer state from an LAS file or from another drawing that are duplicates of layer states in the current drawing, you can choose to overwrite the existing layer state or not import it.

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.

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.

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.

Quick Reference

LAYERSTATE

Saves, restores, and manages named layer states.

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.

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.

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® Colors
- 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

AutoCAD 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.

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 property of the Properties palette when no object is selected. You can also set the current color with 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.

PANTONE® Color Books

Pantone has updated the PANTONE MATCHING SYSTEM® with the PANTONE® PLUS SERIES of Publications that provides a chromatic arrangement of colors. In AutoCAD-based products, the RGB values of the PANTONE Colors

that are assigned to objects are preserved in all current and legacy drawing files.

Color book (.acb) files provide access through the Select Color dialog box to the names of all PANTONE Colors and color books. These .acb files are installed in the \Support\Color folder in the AutoCAD installation folder.

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.

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.

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.

Quick Reference

COLOR

Sets the color for new objects.

CECOLOR

Sets the color of new objects.

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 Color property on the Properties palette when no objects are selected from BYLAYER to a specific color.

See also:

- Override Layer Properties in Viewports on page 529

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.

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.

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.

Quick Reference

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.

CECOLOR

Sets the color of new objects.

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.

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.

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.

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.

Quick Reference

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.

CECOLOR

Sets the color of new objects.

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.

Some linetype definitions include text and symbols.

You can define a custom linetype that will orient the imbedded text to keep it readable automatically.



For more information about controlling text in linetypes, see Text in Custom Linetypes.

See also:

- "Custom Linetypes" in the *Customization Guide*

Quick Reference

LINETYPE

Loads, sets, and modifies linetypes.

REVERSE

Reverses the vertices of selected lines, polylines, splines, and helixes, which is useful for linetypes with included text, or wide polylines with differing beginning and ending widths.

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.

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.

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.

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.

Quick Reference

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.

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.

Set the Current Linetype

All objects are created using the current linetype.

You can set the current linetype with the:

- Linetype control on the Properties panel or toolbar
- Linetype property on the Properties palette
- Linetype Manager

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.

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.

Quick Reference

LINETYPE

Loads, sets, and modifies linetypes.

CELTYPE

Sets the linetype of new objects.

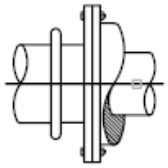
Change the Linetype of an Object

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.

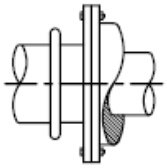
You have three choices for changing the linetype of an object:

- 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 Linetype property on the Properties palette when no objects are selected from BYLAYER to a specific linetype.



object selected



result-continuous linetype changed to center

See also:

- Override Layer Properties in Viewports on page 529

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.

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.

Quick Reference

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.

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.

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.

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.

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.

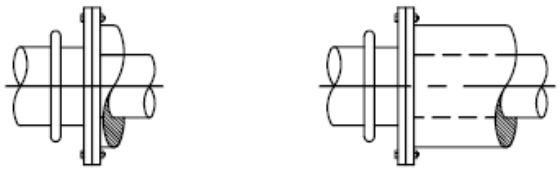
PSLTSCALE

Controls the linetype scaling of objects displayed in paper space viewports.

Display Linetypes on Short Segments and Polylines

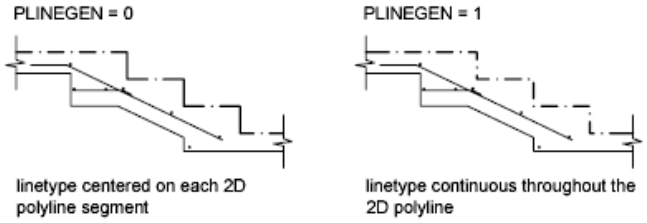
You can center the pattern of a linetype on each segment of a polyline, and you can control how the linetype is displayed on short segments.

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 571.

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.

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.

Quick Reference

PROPERTIES

Controls properties of existing objects.

PLINEGEN

Sets how linetype patterns generate around the vertices of a 2D polyline.

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 Show/Hide Lineweight 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 Show/Hide Lineweight button on the status bar and choosing Settings.

See also:

- Draw Polylines on page 693

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.

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.

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.

Quick Reference

LWEIGHT

Sets the current lineweight, lineweight display options, and lineweight units.

PLINE

Creates a 2D polyline, a single object that is composed of line and arc segments.

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.

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 Show/Hide Lineweight button 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

layout, 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 preview using PREVIEW or PLOT.

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 or the Page Setup Dialog Box.

To display or hide lineweights

Use one of the following methods:

- Click Show/Hide Lineweight on the status bar.
- Select or clear Display Lineweight in the Lineweight Settings dialog box.
- Set the LWDISPLAY system variable to 0 or 1.

Quick Reference

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, a single object that is composed of line and arc segments.

PLOT

Plots a drawing to a plotter, printer, or file.

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.

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. You can set the current lineweight with the:

- Lineweight control on the Properties panel or toolbar
- Lineweight property on the Properties palette
- Lineweight Settings dialog box
- CELWEIGHT system variable

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.

Quick Reference

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, a single object that is composed of line and arc segments.

PLOT

Plots a drawing to a plotter, printer, or file.

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.

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 Lineweight property on the Properties palette when no objects are selected from BYLAYER to a specific lineweight.

See also:

- Override Layer Properties in Viewports on page 529

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.

Quick Reference

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, a single object that is composed of line and arc segments.

PLOT

Plots a drawing to a plotter, printer, or file.

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.

Control the Display Properties of Certain Objects

You can control how overlapping objects and certain other objects are displayed and plotted.

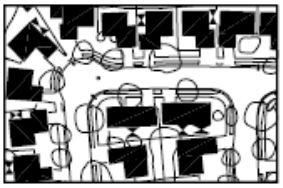
Control the Display of Polylines, Hatches, Gradient Fills, Lineweights, and Text

You can simplify the display of certain kinds of objects in order to speed performance.

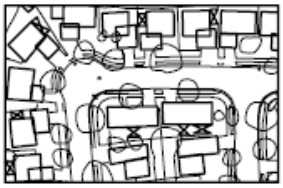
Display performance is improved when wide polylines and donuts, solid-filled polygons (two-dimensional solids), hatches, gradient fills, and text are displayed in simplified form. Simplified display also increases the speed of creating test plots.

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.



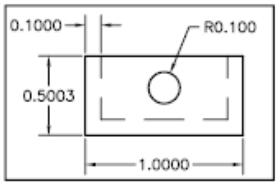
Fill mode on



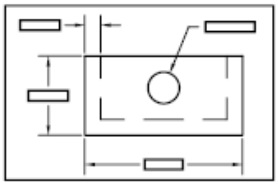
Fill mode off

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 Show/Hide Lineweight button on the status bar. 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 516
- Display Lineweights on page 577
- Use TrueType Fonts on page 1473

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.

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.

To turn lineweights on or off

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 or clear Display Lineweight.
4. Click OK.

Quick Reference

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.

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.

TEXTQLTY

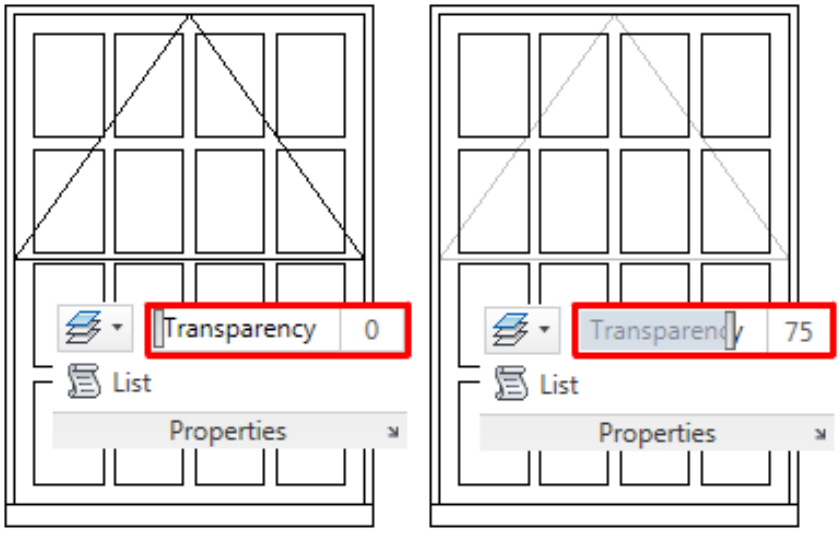
Sets the resolution tessellation fineness of text outlines.

Control the Transparency of Objects

You can control the transparency level of objects and layers.

Set the transparency level of selected objects or layers to enhance drawings or reduce the visibility of areas that are included for reference only.

Transparency can be set to ByLayer, ByBlock, or to a specific value.



no transparent objects **selected objects 75% transparent**

IMPORTANT For perfomance reasons, plotting transparency is disabled by default. To plot transparent objects, check the Plot Transparency option in either the Plot dialog box or Page Setup dialog box.

To control the transparency level of objects

1. Select the objects whose transparency level you want to change.
2. On the Home tab ➤ Properties panel, drag the Transparency slider, or enter a value from 0 to 90 in the Trasparency box.
3. Press Esc to set the transparency level and clear the selection.

TIP You can also control the transparency level from the Properties palette by double clicking an object.

Double clicking hatches or fills displays the Hatch Editor ribbon contextual tab, which contains a transparency control for these objects.

Quick Reference

CHPROP

Changes the properties of an object.

CHANGE

Changes the properties of existing objects.

LAYER

Manages layers and layer properties.

CETRANSPARENCY

Sets the transparency level for new objects.

TRANSPARENCYDISPLAY

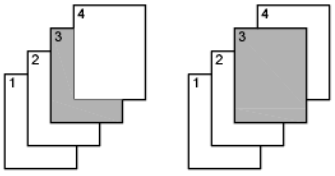
Controls whether the object transparency is displayed.

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. HATCHTOBACK changes the draw order of hatches and gradient and solid fills 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.)

Quick Reference

DRAWORDER

Changes the draw order of images and other objects.

HATCHTOBACK

Sets the draw order for all hatches in the drawing to be behind all 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

Saves selected objects or converts a block to a specified drawing file.

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.

Control the Display of Objects

Control the display of objects by isolating or hiding a selection set.

Use ISOLATEOBJECTS and HIDEOBJECTS to create a temporary drawing view with selected objects isolated or hidden. This saves you the time of having to track objects across layers. If you isolate objects, only the isolated objects appear in the view. Use UNISOLATEOBJECTS to redisplay the objects. When you close and reopen the drawing, all previously hidden objects are displayed. Use the OBJECTISOLATIONMODE system variable to control whether objects remain hidden between drawing sessions.

The Light Bulb icon on the status bar indicates when one or more objects are isolated or hidden. Clicking the Light Bulb icon allows you to access options to isolate, unisolate, or hide objects.

To isolate objects

1. Select the objects you want to isolate.
2. Right-click in the drawing area and select **Isolate ➤ Isolate Objects**.
   Only the selected objects are displayed. All other objects are hidden.
3. To redisplay hidden objects, right-click in the drawing area and select **Isolate ➤ End Object Isolation**.

To hide objects

1. Select the objects you want to hide.
2. Right-click in the drawing area and select **Isolate ➤ Hide Objects**.
   The selected objects are hidden.

3 To redisplay hidden objects, right-click in the drawing area and select **Isolate ➤ End Object Isolation**.

To keep objects hidden or isolated between drawing sessions

1 Before hiding or isolating objects, set OBJECTISOLATIONMODE to 1.

2 Hide or isolate objects in your drawing. When you save, close, and reopen your drawing, the selected objects remain hidden or isolated.

Quick Reference

HIDEOBJECTS

Hides selected objects.

ISOLATEOBJECTS

Displays selected objects across layers; unselected objects are hidden.

UNISOLATEOBJECTS

Displays previously hidden objects.

OBJECTISOLATIONMODE

Controls whether hidden objects remain hidden between drawing sessions.

Use Precision Tools

17

You can use a variety of precision drawing tools to help you produce accurate drawings quickly and without performing tedious calculations.

Work with the User Coordinate System (UCS)

The UCS is the active coordinate system that establishes the XY plane (work plane) and Z-axis direction for drawing and modeling.

Overview of the User Coordinate System (UCS)

The UCS is the active coordinate system that establishes the *XY* plane (*work plane*) and *Z*-axis direction for drawing and modeling. You can set the UCS origin and its *X*, *Y*, and *Z* axes to suit your needs. The UCS is useful in 2D design and essential in 3D design because it controls features that include

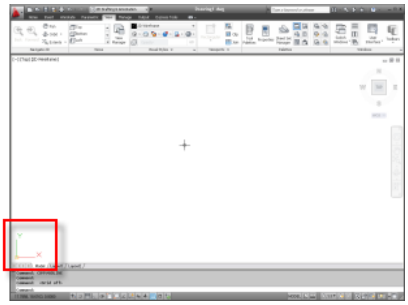
- The *XY* plane (or *work plane*)on which objects are created and modified
- The horizontal and vertical orientation used for features like Ortho mode, polar tracking, and object snap tracking
- The alignment and angle of the grid, hatch patterns, text, and dimension objects
- The origin and orientation for coordinate entry and absolute reference angles
- For 3D operations, the orientation of work planes, projection planes, and the *Z* axis (for vertical direction and axis of rotation)

Understand the World Coordinate System (WCS)

The WCS is a fixed Cartesian coordinate system. Internally, all objects are defined by their WCS coordinates, and the WCS and the UCS are coincident in a new drawing. However, it is usually more convenient to create and edit objects based on the UCS, which can be customized to suit your needs.

Understand the UCS Icon

The UCS icon indicates the location and orientation of the current UCS. You can manipulate the UCS icon using grips. For more information, see The UCS Icon on page 36 and the UCSICON command.

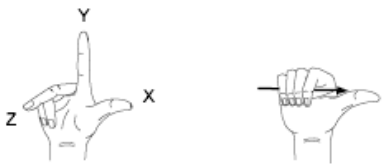


NOTE If the location of the UCS origin is not visible in a viewport, the UCS icon is displayed in the lower-left corner of the viewport instead.

Understand the UCS in 3D

When you create or modify objects in a 3D environment, you can move and reorient the UCS anywhere in 3D space to simplify your work. The UCS is useful for entering coordinates, creating 3D objects on 2D work planes, and rotating objects in 3D.

NOTE The UCS icon follows the traditional right-hand rule in determining positive axis directions and rotation directions.



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.

See also:

- Control the User Coordinate System (UCS) on page 594
- Work with Named UCS Definitions and Preset Orientations on page 596

To restore the UCS to be coincident with the WCS (World Coordinate System)

- Click the UCS origin grip, and then click World.

To restore the previous UCS

- Right-click the UCS icon, and then click Previous.

Quick Reference

Commands

UCS

Sets the origin and orientation of the current UCS.

UCSICON

Controls the visibility, placement, appearance, and selectability of the UCS icon.

UCSMAN

Manages UCS definitions.

System Variables

ELEVATION

Stores the current elevation of new objects relative to the current UCS.

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.

Control the User Coordinate System (UCS)

Customize the UCS origin and orientation using the UCS origin and axes grips, the UCS icon shortcut menu, or the UCS command.

You can align the UCS icon with existing objects, including 3D faces or edges.

To align the UCS with an existing 3D object

1. Click the UCS icon, and then click Move and Align.
2. Drag the UCS icon over the portion of the object with which you want to align it.
3. Click to place the new UCS.

To define a new UCS origin using the origin grip

1. Click the UCS icon.
2. Click and drag the square origin grip to its new location.

The UCS origin (0,0,0) is redefined at the point you specify.

TIP To place the origin with precision, use object snaps or grid snaps, or enter a specific *X*,*Y*,*Z* coordinate.

To specify a new UCS with three points

1. Right-click the UCS icon, and click 3 Point.
2. Specify a new origin point.
3. Specify a point on the positive *X* axis of the new UCS.
4. Specify a point on the positive portion of the *XY* plane of the new UCS.

To rotate the UCS about the X, Y, or Z axis

- Right-click the UCS icon, and click Rotate Axis. Click X, Y, or Z.
  As you drag the cursor, the UCS rotates in the positive direction around the specified axis (see Apply the Right-Hand Rule). You can also specify a rotation angle.

TIP Hover over the grips at the ends of the *X*, *Y*, and *Z* axes to access the rotate options.

To change the orientation of the Z axis

1. Right-click the UCS icon, and click Z Axis.
2. Specify a point for the new origin (0,0,0).
3. Specify a point that lies on the positive *Z* axis.

Quick Reference

Commands

UCS

Sets the origin and orientation of the current UCS.

UCSICON

Controls the visibility, placement, appearance, and selectability of the UCS icon.

UCSMAN

Manages UCS definitions.

System Variables

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.

Work with Named UCS Definitions and Preset Orientations

Create and save as many UCS definitions as you need. Each UCS definition can have its own origin and X, Y, and Z axes. You can also choose from several preset orientations.

See also:

- Overview of the User Coordinate System (UCS) on page 591
- Control the User Coordinate System (UCS) on page 594

NOTE By default, the Coordinates panel is hidden in the Drafting & Annotation workspace. To display the Coordinates panel, click the View tab, then right-click and choose Show Panels, and then click Coordinates. In 3D workspaces, the Coordinates panel is on the Home tab.

To save a UCS definition

1. Right-click the UCS icon, and click Named UCS ➤ Save.
2. Enter a name.

   You can use up to 255 characters, including letters, digits, and the special characters dollar sign ($), hyphen (-), and underscore (_).

To restore a named UCS definition

- Right-click the UCS icon, click Named UCS, and then click the UCS definition you want to restore.

To select a preset UCS orientation

1. Click View tab ➤ Coordinates 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.

To rename a UCS definition

1. Click View tab ➤ Coordinates panel ➤ Named UCS. 
2. In the UCS dialog box, Named UCSs tab, right-click the UCS definition you want to rename, and click Rename.
3. Enter a new name.
4. Click OK.

To delete a UCS definition

1. Click View tab ➤ Coordinates panel ➤ Named UCS. 
2. In the UCS dialog box, Named UCSs tab, select the UCS definition you want to delete.
3. Press Delete.

   You cannot delete the current UCS or a UCS with the default name UNNAMED.

Quick Reference

Commands

UCS

Sets the origin and orientation of the current UCS.

UCSMAN

Manages UCS definitions.

UCSICON

Controls the visibility, placement, appearance, and selectability of the UCS icon.

System Variables

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.

UCSVP

Determines whether the UCS in viewports remains fixed or changes to reflect the UCS of the current viewport.

UCSYDIR

Stores the Y direction of the current UCS for the current viewport in the current space.

Assign UCS Definitions to Viewports

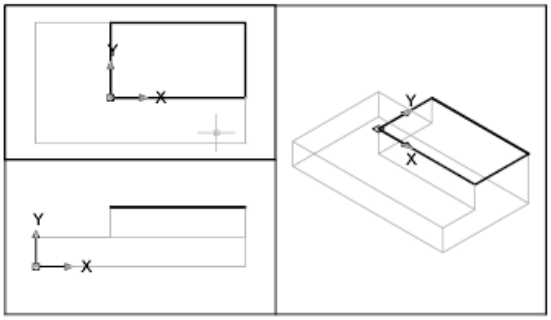
To facilitate editing objects in different views, you can define a different UCS (User Coordinate System) for each view.

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 create a different UCS definition 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.

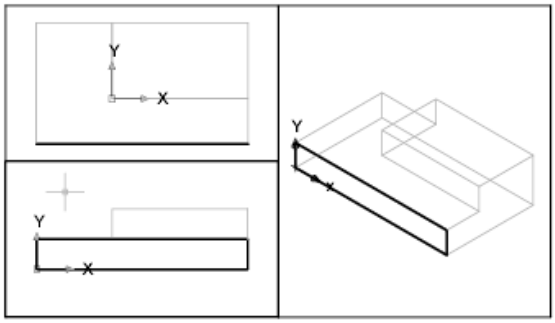
The example is illustrated in the following figures. The first figure shows the isometric viewport reflecting the UCS of the upper-left, or top, viewport, which is current.



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 earlier releases, you can set the value of the UCSVP system variable to 0 in all active viewports.

Quick Reference

Commands

UCS

Sets the origin and orientation of the current UCS.

System Variables

UCSNAME

Stores the name of the current coordinate system for the current viewport in the current space.

UCSVP

Determines whether the UCS in viewports remains fixed or changes to reflect the UCS of the current viewport.

UCSFOLLOW

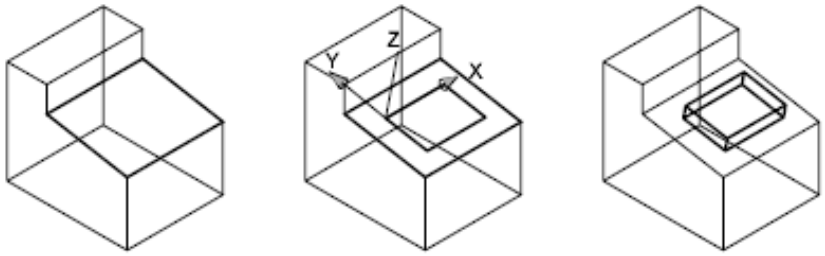
Generates a plan view whenever you change from one UCS to another.

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.



face selected | base point and origin of dynamic ucs | result

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

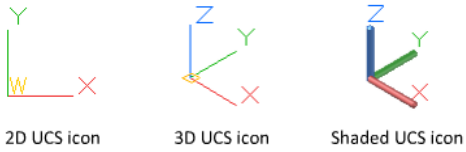
Controls whether dynamic UCS acquisition is active or not.

Control the Display of the User Coordinate System Icon

The user coordinate system icon (UCS icon) helps you visualize the current orientation of the UCS. 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 a 2D or 3D style of the icon to represent the UCS when working in 2D environment. Shaded style of icon is displayed to represent the UCS in the 3D environment.



2D UCS icon 3D UCS icon Shaded UCS icon

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.

The UCS Icon and Multiple Viewports

If you have multiple viewports, each viewport displays its own UCS icon.

Display and Hide the UCS Icon

In some circumstances, you might need to hide the UCS icon. With the UCSICON command, you can turn off the UCS icon in a single viewport or all viewports. Each layout also provides a UCS icon in paper space.

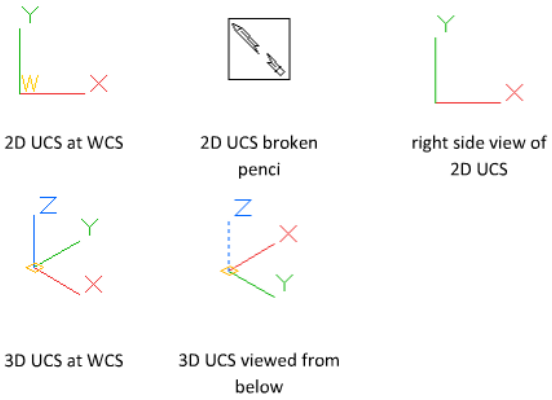
You can also hide the UCS icon based on the current *visual style*, and whether perspective is turned on. Three system variables are available:

- Use UCS2DDISPLAYSETTING to hide the UCS icon when the current visual style is 2D Wireframe.
- Use UCS3DPARADISPLAYSETTING to hide the UCS icon when perspective (PERSPECTIVE) is turned off in a viewport with a 3D visual style.
- Use UCS3DPERPDISPLAYSETTING to hide the UCS icon when perspective is turned on in a viewport with a 3D visual style.

NOTE These system variables take precedence over the UCSICON setting in viewports with certain visual styles.

Variations in UCS Icon Types

The UCS icon is displayed in various ways to help you visualize the orientation of the work plane. 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, 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.

NOTE By default, the Coordinates panel is hidden in the Drafting & Annotation workspace. To display the Coordinates panel, click the View tab, then right-click and choose Show Panels, and then click Coordinates. In 3D workspaces, the Coordinates panel is on the Home tab.

To turn the display of the UCS icon on and off

- Click View tab ➤ Coordinates panel ➤ Toggle Icon. 
  The check mark indicates whether the icon is on or off.

To display the UCS icon at the UCS origin

- Click View tab ➤ Coordinates 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.

To change the appearance of the UCS icon

1. Click View tab ➤ Coordinates panel ➤ UCS Icon Properties.
2. In the UCS Icon dialog box, change the settings.
3. Click OK.

Quick Reference

Commands

UCSICON

Controls the visibility, placement, appearance, and selectability of the UCS icon.

VSCURRENT

Sets the visual style in the current viewport.

System Variables

UCS2DDISPLAYSETTING

Displays the UCS icon when the 2D Wireframe visual style is current.

UCS3DPARADISPLAYSETTING

Displays the UCS icon when perspective view is off and a 3D visual style is current.

UCS3DPERPDISPLAYSETTING

Displays the UCS icon when perspective view is on and a 3D visual style is current.

PERSPECTIVE

Specifies whether the current viewport displays a perspective view.

UCSICON

Displays the UCS icon for the current viewport or layout.

Enter Coordinates to Specify Points

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.

Overview of Coordinate Entry

You can enter two-dimensional coordinates as either Cartesian (*X,Y*) or polar coordinates. When dynamic input is on, you can enter coordinate values in tooltips near the cursor.

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 *work plane*. The work plane 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 609
- Enter Polar Coordinates on page 611

- Enter 3D Coordinates on page 614
- Use Dynamic Input on page 619

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 display the coordinate values of a point

1. Click Home tab ➤ Utilities panel ➤ ID Point. 
2. Select the location you want to identify.
   The *X,Y,Z* coordinate values are displayed at the Command prompt.

To visually locate a point

1. Click Home tab ➤ Utilities panel ➤ ID Point. 
2. At the Command prompt, enter the coordinate values of the point you want to locate.

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+I.
- Set the COORDS system variable to 0 for static display, 1 for dynamic display, or 2 for distance and angle display.

Quick Reference

Commands

ID

Displays the UCS coordinate values of a specified location.

LIST

Displays property data for selected objects.

System Variables

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.

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.

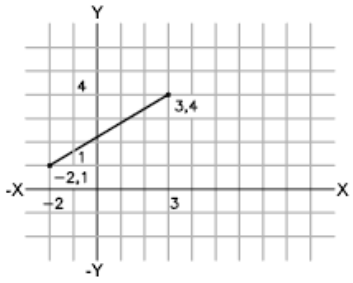
Absolute coordinates are based on the UCS origin (0,0), which is the intersection of the *X* and *Y* axes. Use absolute coordinates when you know the precise *X* and *Y* values of the point.

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 619.

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:

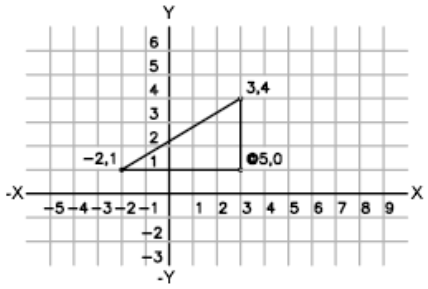


Relative coordinates are based on the last point entered. Use relative coordinates when you know the location of a point in relation to the previous point.

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

System Variables

COORDS

Controls the format and update frequency of coordinates on the status line.

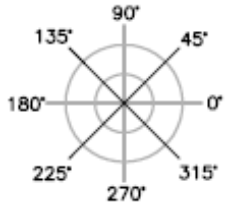
Enter Polar Coordinates

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.

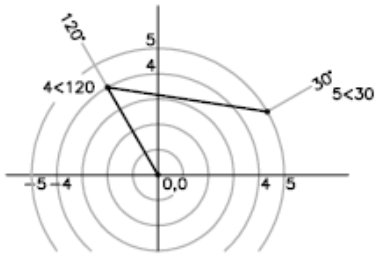


Absolute polar coordinates are measured from the UCS origin (0,0), which is the intersection of the *X* and *Y* axes. Use absolute polar coordinates when you know the precise distance and angle coordinates of the point.

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 619.

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**

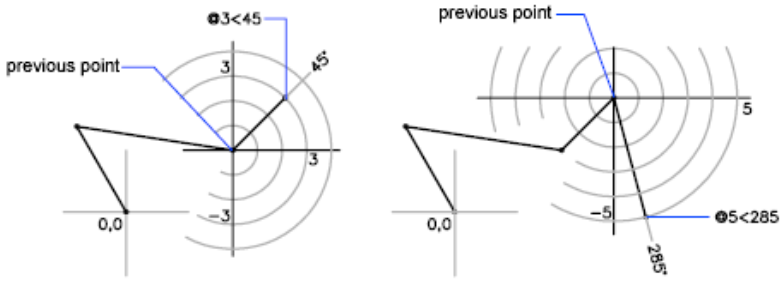


Relative coordinates are based on the last point entered. Use relative coordinates when you know the location of a point in relation to the previous point.

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.

Enter 3D Coordinates

Cartesian, cylindrical, or spherical coordinates locate points when you are creating objects in 3D.

Enter 3D Cartesian Coordinates

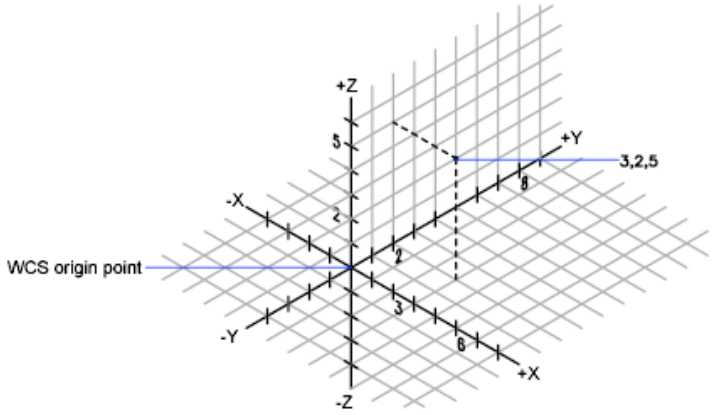
3D Cartesian coordinates specify a precise location by using three coordinate values: *X*, *Y*, and *Z*.

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

Sets the origin and orientation of the current UCS.

UCSICON

Controls the visibility, placement, appearance, and selectability 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.

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.

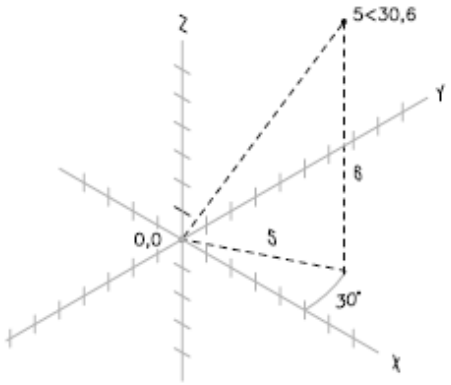
Cylindrical coordinate entry is the 3D equivalent of 2D polar coordinate entry. It specifies an additional coordinate on an axis that is perpendicular to the

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.

Enter Spherical Coordinates

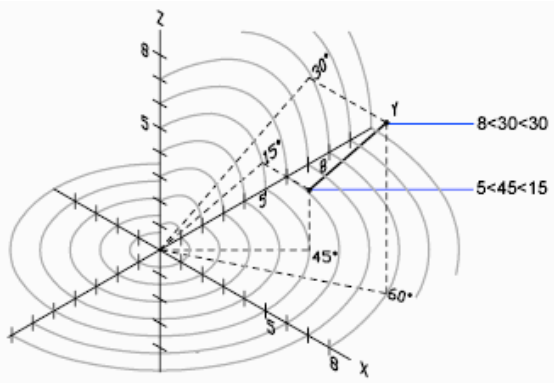
3D spherical coordinates specify a location by a distance from the origin of the current UCS, an angle from the *X* axis in the *XY* plane, and an angle from the *XY* plane.

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.

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 line. You can hide the command line 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

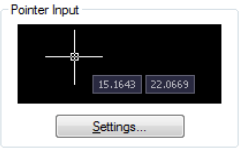
Click the dynamic input button  on the status bar to turn dynamic input on and off. Dynamic input has three components: pointer input, dimensional input, and dynamic prompts. Right-click the dynamic input button and click Settings to control what is displayed by each component when dynamic input is on.

NOTE You can turn it off temporarily by holding down the F12 key.

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.

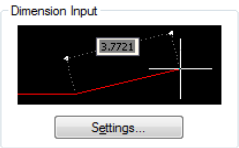
The default for second and subsequent points is relative polar coordinates (relative Cartesian for RECTANG). There is no need to type the at sign (@). If you want to use absolute coordinates, use the pound sign (#) prefix. For example, to move an object to the origin, for the second point prompt, enter **#0,0**.



Use the pointer input settings to change the default format for coordinates and to control when pointer input tooltips are displayed.

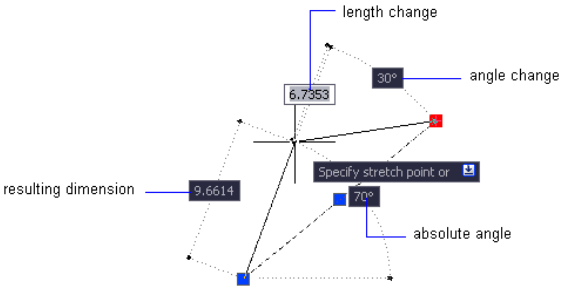
Dimensional Input

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 paste text into 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 change the color, size, or transparency of tooltips

1. Click Tools menu ➤ Drafting 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.

To change pointer input settings

1 Click Tools menu ➤ Drafting 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.

To change dimensional input settings

1 Click Tools menu ➤ Drafting 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.

To display prompts in tooltips

1 Click Tools menu ➤ Drafting 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.

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.

Snap to Locations on Objects (Object Snaps)

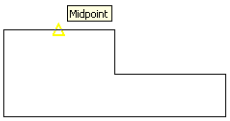
Instead of entering coordinates, you can specify points relative to existing objects such as endpoints of lines or center points of circles.

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
- When prompted for a point, right-click and choose an object snap from the Snap Overrides sub-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 object snap 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.

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.

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.

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. You can also right-click while being prompted for a point and click Snap Overrides.

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:

- Use Object Snaps on page 626
- "Pull-down and Shortcut Menus" in the *Customization Guide*

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.

Quick Reference

Commands

OSNAP

Sets running object snap modes.

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.
- **2D and 3D AutoSnap Marker Colors.** Changes the color of the markers.
- **AutoSnap Marker Size.** Adjusts the size of the marker.

3 Click OK.

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

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.

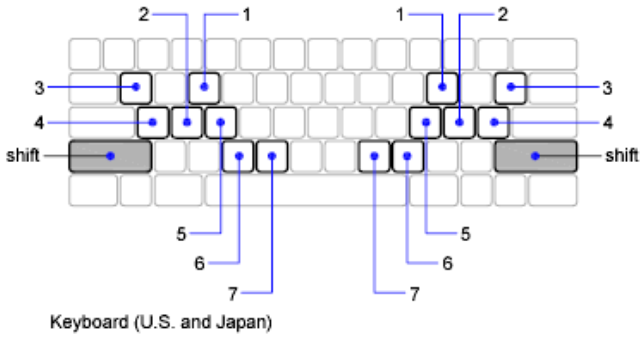
For example, if you have set running object snaps but you want to turn them off for one point, you can hold down F3. When you release this override key, running object snaps are restored.

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.

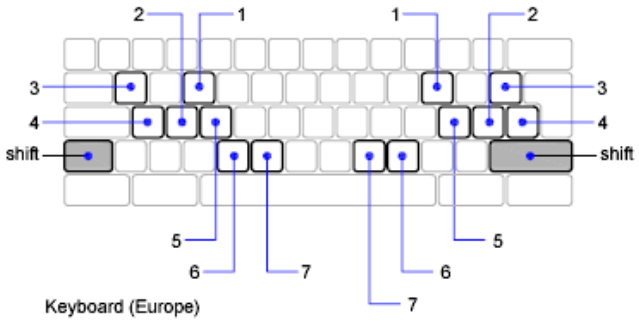
The keys in the following illustration are the default keys, but you can change key assignments and add your own as needed.

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.

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.

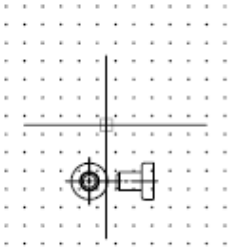
The grid is a rectangular pattern of dots or lines that extends over the area you specify as the grid limits. Using the grid is similar to placing a sheet of grid paper under a drawing. The grid helps you align objects and visualize the distances between them. The grid is not plotted.

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.

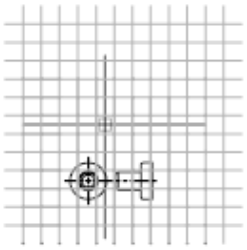
Grid mode and Snap mode are independent but are often turned on at the same time.

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 lines for all visual styles. The grid displays dots only when the current visual style is set to 2D Wireframe. By default, a lined grid is displayed while working in both 2D and 3D. There are several methods to change the current visual style, including the VSCURRENT



Visual styles = 2D wireframe



Visual styles = hidden

command.

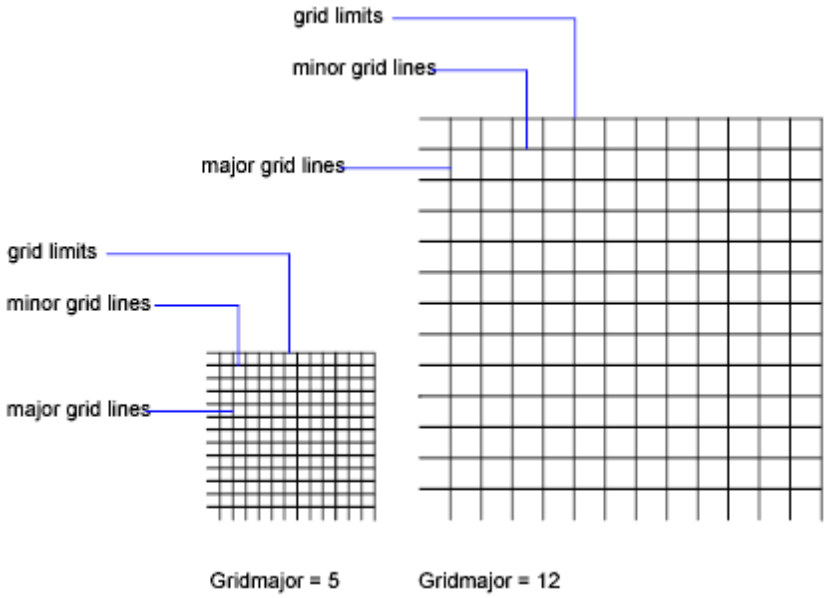
By default, the *X* and *Y* axes of the UCS display in a different color than the grid lines. You can control the color in the Drawing Window Colors dialog box. This dialog box is accessible from the Drafting tab in the Options dialog box.

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.

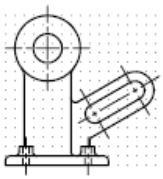
Change Grid and Snap Spacing

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.

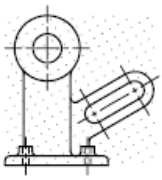
Snap spacing does not have to match grid spacing. For example, you might set a wide grid spacing to be used as a reference but maintain a closer snap spacing for accuracy in specifying points.

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.



UCS rotated 0 degrees



UCS rotated 30 degrees

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 633
- Use a Visual Style to Display Your Model on page 294

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.

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.

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.

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 ➤ Coordinates panel ➤ Z. 
2. Enter the rotation angle for the UCS.
3. Click View tab ➤ Coordinates panel ➤ Origin. 
4. Specify a new origin point for the UCS.
5. Click OK.

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.

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.

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 named layout.

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.

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 643.

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 633

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.



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.

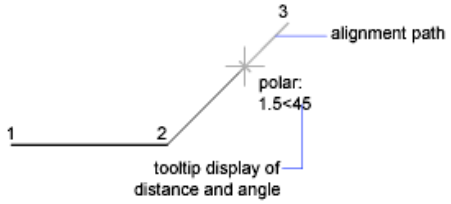
Use Polar Tracking and PolarSnap

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 (UNITS).

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.

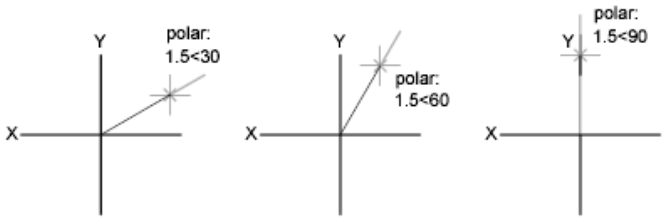


As you move your cursor, alignment paths and tooltips are displayed when you move the cursor near polar angles. The default angle measurement is 90 degrees. Use the alignment path and tooltip to draw your object. You can use polar tracking with Intersection and Apparent Intersection object snaps to find 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. 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 633

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.

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.

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.

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.

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.

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.

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.

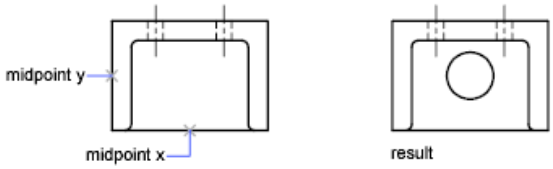
Coordinate filters specify a new coordinate location by using the *X* value from one location, the *Y* value of a second location, and, for 3D coordinates, the *Z* value of a third location. When used with object snaps, coordinate filters extract coordinate values from an existing object.

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:

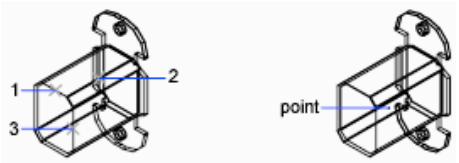
Command: **circle**
Specify center point for circle or [3P/2P/Ttr (tangent tangent radius)]: **.x**
of: **mid**
of: *Select the horizontal line on the lower edge of the holding plate*
of: (need YZ): **mid**
of: *Select the vertical line on the left side of the holding plate*
of: Diameter/<Radius> *Specify 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

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 633.

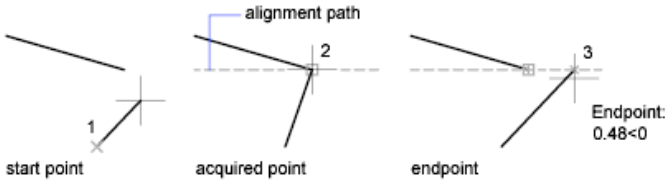
Object snap tracking works in conjunction with object snaps. You must set an object snap before you can track from an object's snap point.

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.

NOTE You can track Perpendicular or Tangent object snap from the last picked point in a command even if the object snap tracking is off.

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.

NOTE 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.

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.

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.

Quick Reference

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 and points using the AutoCAD calculator (CAL).

Both the CAL command and the QuickCalc calculator can be used transparently to calculate distances and points for commands that require input. For more information, see Use the QuickCalc Calculator on page 670.

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 643
- Lock an Angle for One Point (Angle) on page 648

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.

Quick Reference

Commands

LINE

Creates straight line segments.

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

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:

- Work with Blocks on page 915

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

Saves selected objects or converts a block to a specified drawing file.

System Variables

PDMODE

Controls how point objects are displayed.

PDSIZE

Sets the display size for point objects.

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 with the Point Style dialog box (DDPTYPE). 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.

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.

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.

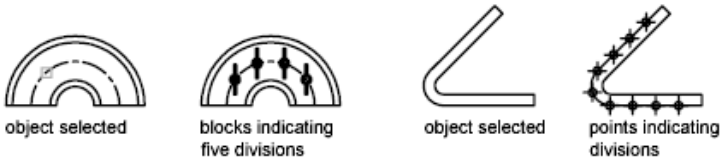
PDSIZE

Sets the display size for point objects.

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 with the Point Style dialog box (DDPTYPE). 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.

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.

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

Saves selected objects or converts a block to a specified drawing file.

System Variables

PDMODE

Controls how point objects are displayed.

PDSIZE

Sets the display size for point objects.

Extract Geometric Information from Objects

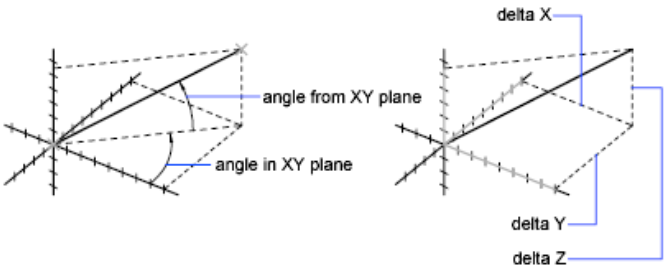
The inquiry and calculation commands can provide information about objects in your drawing and do useful calculations.

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

To calculate the distance and angle between two points

1. Click Home tab ➤ Utilities panel ➤ Measure drop-down ➤ 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.

To calculate the distance and angle between multiple points

1. Click Home tab ➤ Utilities panel ➤ Measure drop-down ➤ 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.

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.

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

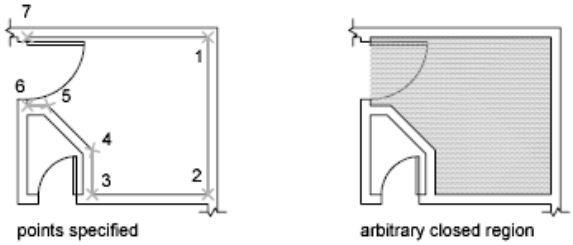
With the MEASUREGEOM and AREA commands, you can specify a series of points or select an object to calculate area. If you need to calculate the combined area of multiple objects, you can keep a running total as you add or subtract one area at a time from the selection set. You cannot use window selection 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.



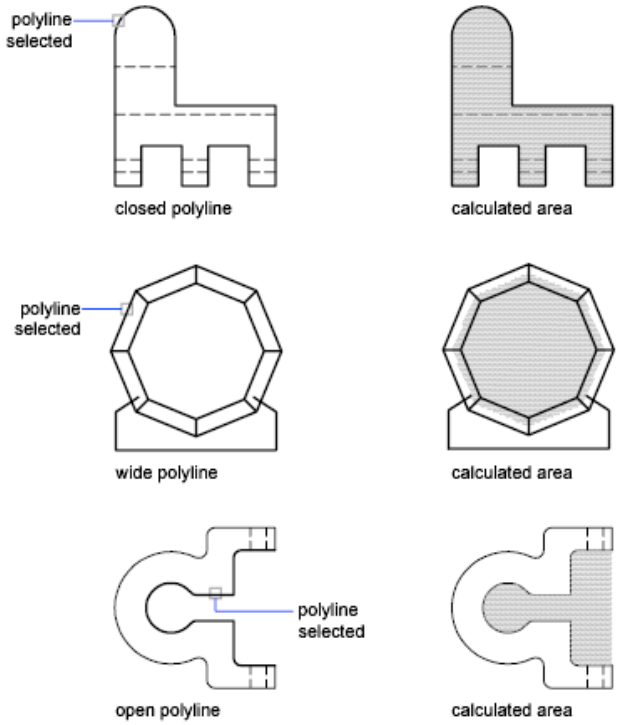
Calculate the Area, Perimeter, or Circumference of an Object

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 calculated 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

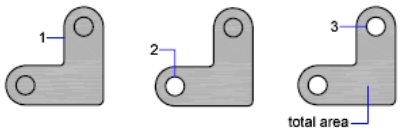
You can subtract more than one area from a combined area as you calculate. For example, if you have calculated the area of a floor plan, you can subtract the area of a room.

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 CAL command or QuickCalc calculator to convert from one system of area units to another. For more information, see Use the QuickCalc Calculator on page 670.

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 733
- Overview of Object Properties on page 507

To calculate an area you define

1. Click Home tab ➤ Utilities panel ➤ Measure drop-down ➤ 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.

To calculate the area of an object

1. Click Home tab ➤ Utilities panel ➤ Measure drop-down ➤ 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.

To add an area as you calculate

1. Click Home tab ➤ Utilities panel ➤ Measure drop-down ➤ 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.

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.

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.

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 CAL command at Command prompt transparently while a command is active or not.

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

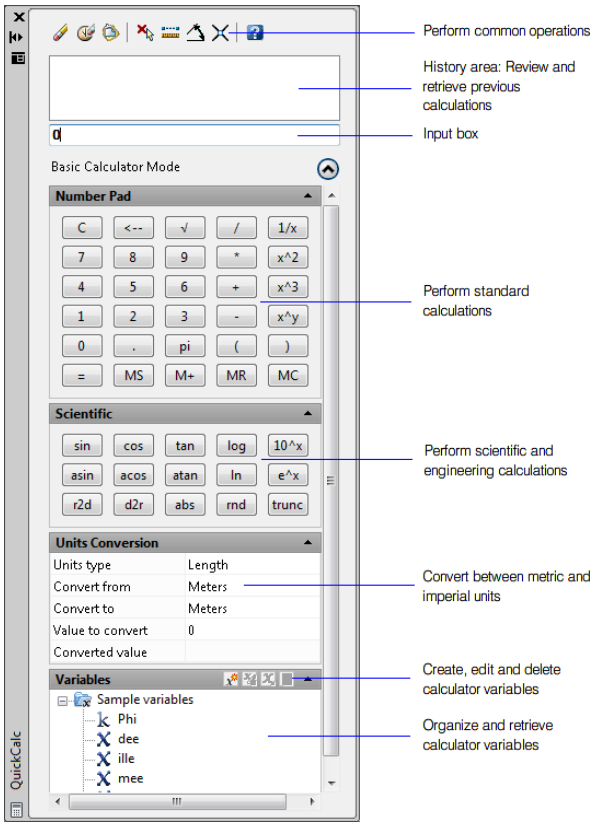
The QuickCalc calculator includes basic features similar to most standard mathematical calculators.

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 149.

Quick Reference

Commands

QUICKCALC

Opens the QuickCalc calculator.

ADCCLOSE

Closes the QuickCalc calculator.

System Variables

QCSTATE

Indicates whether the QuickCalc calculator is open or closed.

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. 
- Right-click the drawing area to display a shortcut menu. Click 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.

Quick Reference

Commands

QUICKCALC

Opens the QuickCalc calculator.

ADCCLOSE

Closes the QuickCalc calculator.

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 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.

To modify a property in the Properties palette with QuickCalc

1. Click View tab ➤ Palettes panel ➤ Properties.
2. 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.
3. 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.

4 Click the calculator icon.

QuickCalc opens and displays the current value of the object in the Input box.

5 Perform a calculation on the displayed value and click the equal sign (=) button.

The new value is displayed in the Input box.

6 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.

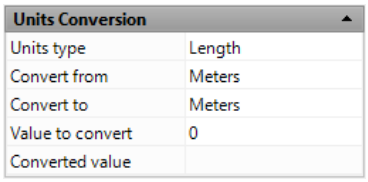
ADCCLOSE

Closes the QuickCalc calculator.

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.

ADCCLOSE

Closes the QuickCalc calculator.

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.

Quick Reference

Commands

ADCCLOSE

Closes the QuickCalc calculator.

QUICKCALC

Opens the QuickCalc calculator.

System Varibles

CALCINPUT

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

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

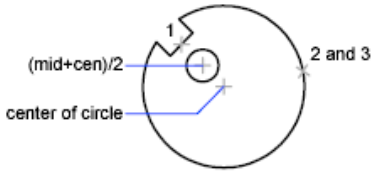
Operator	Operation
( )	Groups expressions
^	Indicates numeric exponent
*, /	Multiplies and divides numbers
+, -	Adds and subtracts numbers

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.

Create Objects 18

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 507.

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:

- Enter Coordinates to Specify Points on page 606
- Use Object Snaps on page 626
- Adjust Grid and Grid Snap on page 635
- Draw Polylines on page 693
- Offset an Object on page 815
- Break and Join Objects on page 843
- Infer Geometric Constraints on page 885

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.

Quick Reference

Commands

LINE

Creates straight line segments.

RAY

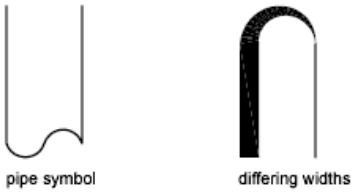
Creates a line that starts at a point and continues to infinity.

XLINE

Creates a line of infinite length.

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.



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.



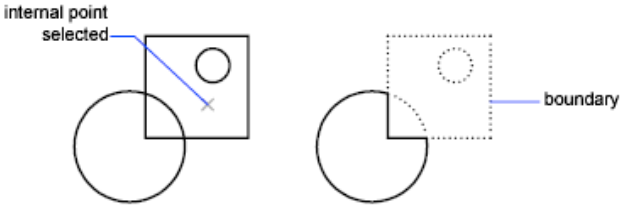
The Width and Halfwidth options set the width of the next polyline segments you draw. Widths greater than zero produce wide lines, which are filled if Fill mode is on and outlined if Fill mode is off.

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 698
- "Modify Complex Objects" on page 79
- Break and Join Objects on page 843
- Control Lineweights on page 574

■ Infer Geometric Constraints on page 885

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.

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.

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.

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.

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, a single object that is composed of line and arc segments.

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 HATCH and BOUNDARY.

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.

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.

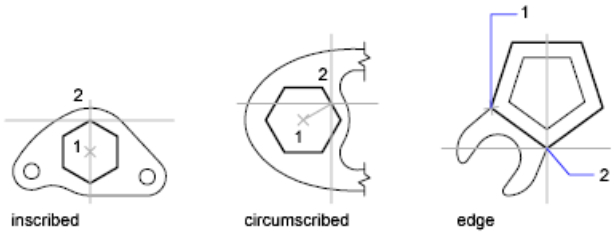
If necessary, you can use EXPLODE to convert the resulting polyline object into lines.

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.

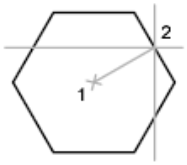


See also:

- Draw Polylines on page 693
- Infer Geometric Constraints on page 885

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).



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.

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.

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.

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 HATCH and BOUNDARY.

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.

Draw Multiline Objects

Multilines are composed of 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.

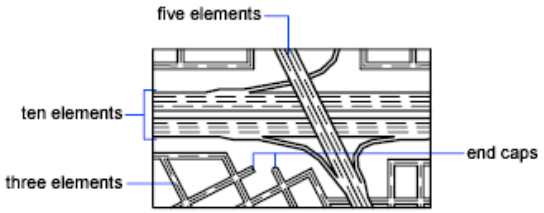
Multiline justification determines which side of the cursor that the multiline is drawn, or whether it is centered on the cursor.

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



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 860

To draw a multiline



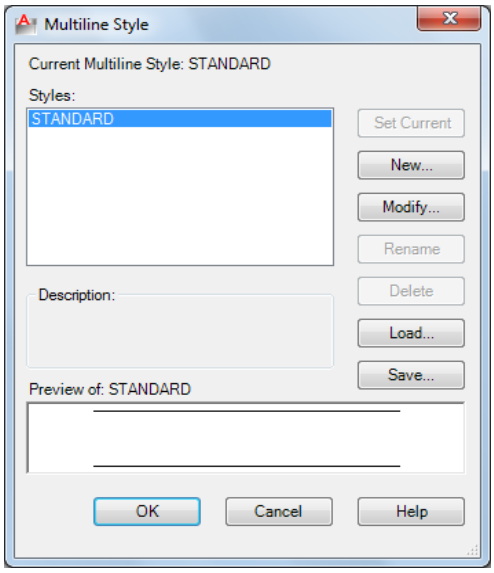
1. Click Draw menu ➤ Multiline.
2. At the Command prompt, enter **st** to select a style.
3. To list available styles, enter the style name or enter **?**.
4. To justify the multiline, enter **j** and select top, zero, or bottom justification.
5. To change the scale of the multiline, enter **s** and 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.

To create a multiline style

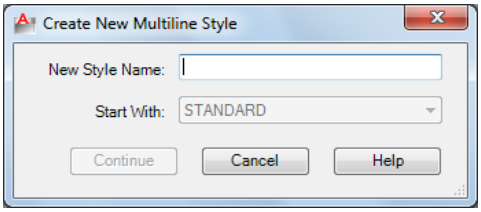


1. Click Format menu ➤ Multiline Style.



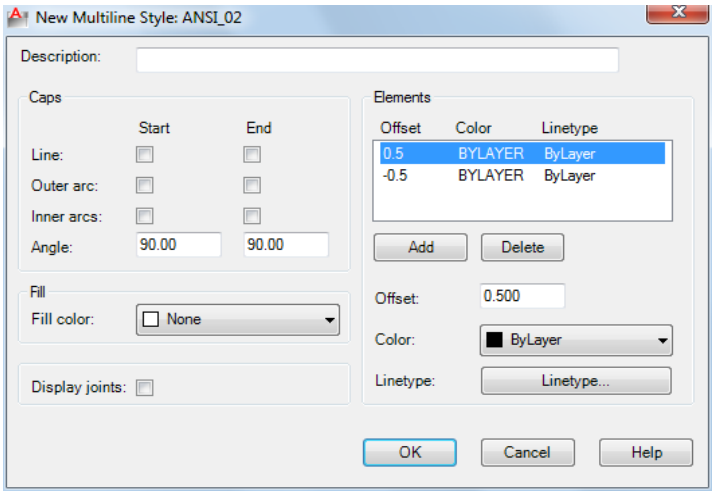
2 In the Multiline Style dialog box, click New.

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.

Draw Freehand Sketches

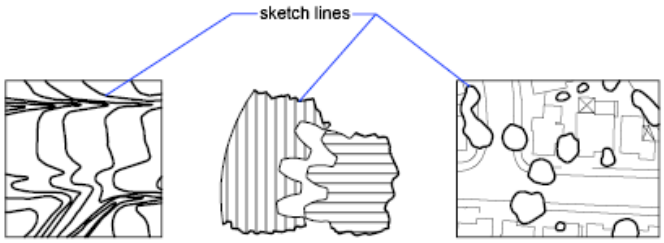
Sketching is useful for creating irregular boundaries or for tracing with a digitizer.

Draw freehand sketches with the SKETCH command. Freehand sketches comprise many line segments that are converted into a line, polyline, or spline.

For Splines, you can determine how closely the spline's curve fits to the freehand sketch.

For any sketch type, set the minimum length (increment) of the line segments. Small line segments allow greater accuracy, but they can greatly increase the drawing file size.

Before sketching, check the CELTYPE system variable to make sure the current linetype is BYLAYER. When you sketch with dot or dash linetypes, smaller line segments can become invisible.



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 cannot 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 cannot select the menus while Tablet mode is on. See your digitizer documentation for details.

To draw freehand sketches

1. At the Command prompt, enter **sketch** and press Enter.
2. Press Enter again to accept the last saved type, increment, and tolerance values.
3. Click in the drawing area to begin sketching.
4. As you move the pointing device, freehand line segments of the specified length are drawn. SKETCH does not accept coordinate input. During the command, freehand lines are displayed in a different color.
5. Click the endpoint to suspend sketching, allowing you to move around the screen without drawing. Click a new start point to resume drawing from the new cursor position.
6. Press Enter to complete the sketch.

Quick Reference

Commands

SKETCH

Creates a series of freehand line segments.

System Variables

DIGITIZER

Identifies digitizers connected to the system.

MAXTOUCHES

SKETCHINC

Sets the record increment for the SKETCH command.

SKPOLY

Determines whether the SKETCH command generates lines, polylines, or splines.

SKTOLERANCE

Draw Curved Objects

Curved objects are arcs, circles, polyline arcs, donuts, ellipses, and splines.

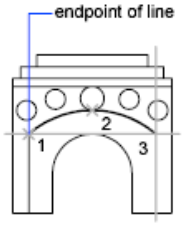
Draw Arcs

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.

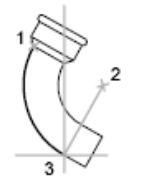


Draw Arcs by Specifying Start, Center, End

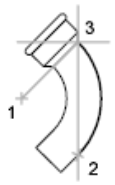
You can create an arc using a start point, center, and a third point that determines the endpoint.

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)



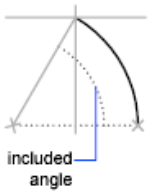
center(1), start(2), end(3)

Draw Arcs by Specifying Start, Center, Angle

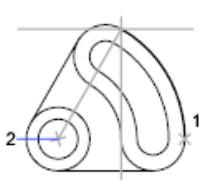
You can create an arc using a start point, center, and an included angle.

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.

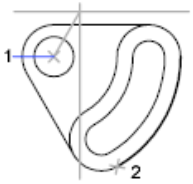
Using different options, you can specify either the start point first or the center point first.



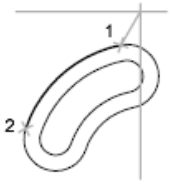
The included angle determines the endpoint of the arc. Use the Start, End, Angle method when you know both endpoints but cannot snap to a center point.



start, center, angle



center, start, angle



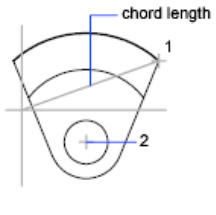
start, end, angle

Draw Arcs by Specifying Start, Center, Length

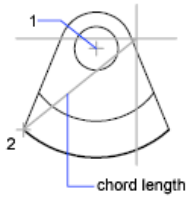
You can create an arc using a start point, center, and the length of a chord.

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.



start, center, length



center, start, length

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

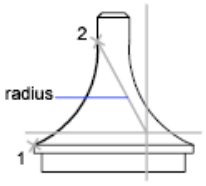
You can create an arc using a start point, endpoint, and a tangent direction at the start point.

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.

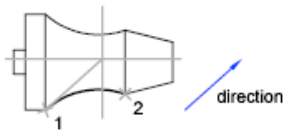
Draw Arcs by Specifying Start, End, Radius

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.



start, end, radius



start, end, direction

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 693
- Break and Join Objects on page 843

To draw an arc by specifying three points

1. Click Home tab ➤ Draw panel ➤ Arc drop-down ➤ 3-Point. 
2. Specify the start point.
3. Specify a point on the arc.
4. Specify the endpoint.

To draw an arc using a start point, a center point, and an endpoint

1. Click Home tab ➤ Draw panel ➤ Arc drop-down ➤ Start, Center, End.



2. Specify a start point.
3. Specify the center point.
4. Specify the endpoint.

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.

To continue an arc with a tangential arc

1. Complete the arc.
2. Click Home tab ➤ Draw panel ➤ Arc drop-down ➤ Continue. 
3. Specify the second endpoint of the tangent 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.

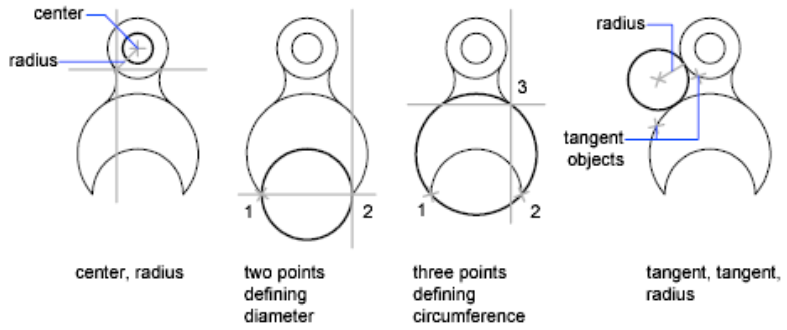
WHIPARC

Controls whether the display of circles and arcs is smooth.

Draw Circles

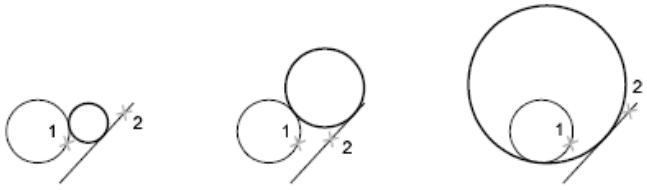
To create circles, you can specify various combinations of center, radius, diameter, points on the circumference, and points on other objects.

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.



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 626
- Draw Isometric Circles on page 2141

To draw a circle by specifying a center point and radius or diameter

1. Do one of the following:
   - Click Home tab ➤ Draw panel ➤ Circle drop-down ➤ Center, Radius. 
   - Click Home tab ➤ Draw panel ➤ Circle drop-down ➤ Center, Diameter. 
2. Specify the center point.
3. Specify the radius or diameter.

To create a circle tangent to two objects

1. Click Home tab ➤ Draw panel ➤ Circle drop-down ➤ 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.

Quick Reference

Commands

CIRCLE

Creates a circle.

OFFSET

Creates concentric circles, parallel lines, and parallel curves.

System Variables

CIRCLERAD

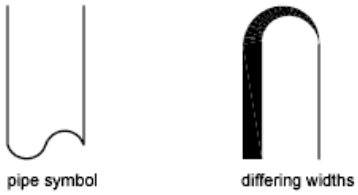
Sets the default circle radius.

WHIPARC

Controls whether the display of circles and arcs is smooth.

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.

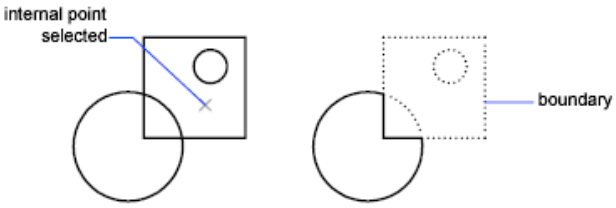
Taper

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 853
- Modify Polylines on page 846
- Break and Join Objects on page 843
- Control Lineweights on page 574
- Infer Geometric Constraints on page 885

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.

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.

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.

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.

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, a single object that is composed of line and arc segments.

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 HATCH and BOUNDARY.

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.

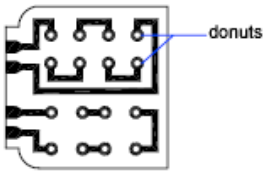
PLINEWID

Stores the default polyline width.

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.



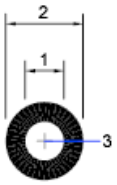
filled rings



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

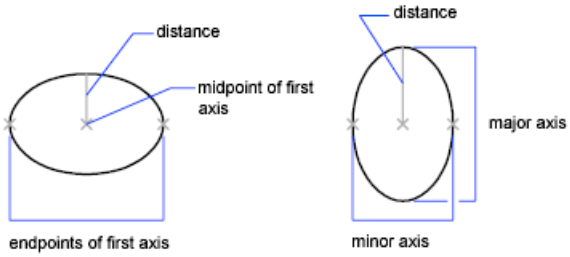
Sets the default for the outside diameter of a donut.

FILLMODE

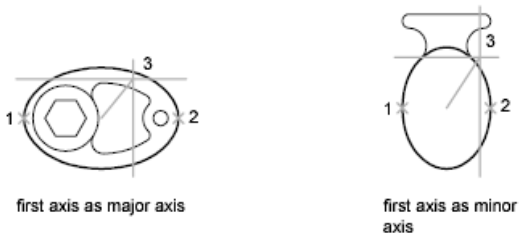
Specifies whether hatches and fills, 2D solids, and wide polylines are filled in.

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 (DSETTINGS command).

See also:

- Draw Isometric Circles on page 2141
- Break and Join Objects on page 843

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 ➤ Ellipse drop-down ➤ Axis, End.



4 Enter **i** (Isocircle).

5 Specify the center of the circle.

6 Specify the radius or diameter of the circle.

To draw a true ellipse using endpoints and distance

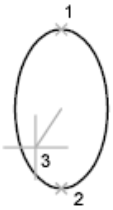


1 Click Home tab ➤ Draw panel ➤ Ellipse drop-down ➤ Axis, End.

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.



To draw an elliptical arc using start and end angles

1 Click Home tab ➤ Draw panel ➤ Ellipse drop-down ➤ Elliptical Arc.



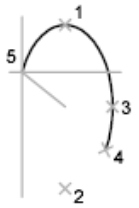
2 Specify endpoints for the first axis (1 and 2).

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.



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.

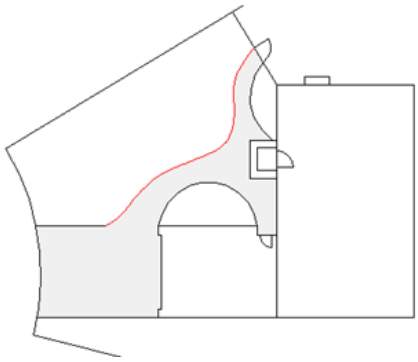
Draw Splines

A spline is a smooth curve that passes through or near a set of points that influence the shape of the curve.

SPLINE and BLEND create curves called nonuniform rational B-splines (NURBS), referred to as *splines* for simplicity.

By default, a spline is a series of blended curve segments of degree 3 (also called cubic) polynomials. Cubic splines are the most common, and mimic the splines that are created manually using flexible strips that are shaped by weights at data points.

In the following example, SPLINE was used to create the highlighted boundary of the concrete walkway.

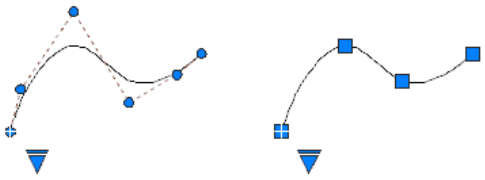


BLEND was used to create splines between lines and arcs for a golf course design. The resulting splines are tangent to the selected lines and curves without changing the lengths of the selected objects.

Splines are also used for creating solids and surfaces for 3D modeling. For more information, see Create Solids and Surfaces from Lines and Curves on page 1030.

Understand Control Vertices and Fit Points

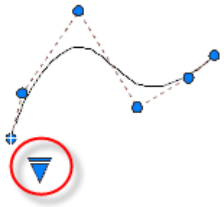
You can create or edit splines using either *control vertices*, or *fit points*. The spline on the left displays control vertices along a *control polygon*, and the spline on the right displays fit points.



The options available in SPLINE depend on which method is used to create the spline.

CVSHOW and CVHIDE determine whether the control vertices are displayed on a spline even when the spline is not selected.

Use the triangular grip on a selected spline to switch between displaying control vertices and displaying fit points. You can use the round and square grips to modify a selected spline. For more information, see Modify Splines on page 853.



IMPORTANT Switching the display from control vertices to fit points automatically changes the selected spline to degree 3. Splines originally created using higher-degree equations will likely change shape as a result.

Create Splines Using Control Vertices

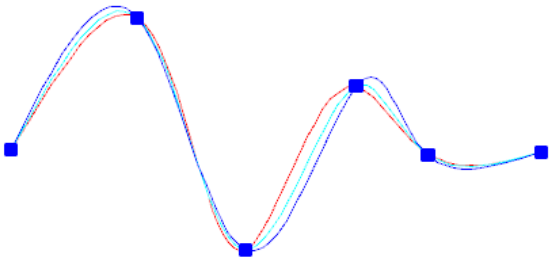
When you create splines using control vertices, the points you specify display temporary lines between them, forming a control polygon that determines the shape of the spline.

The advantage of changing the shape of a spline using control vertices is the fine control this method provides. With this method, you can also specify lower or higher degree polynomials, including degree 1 (linear), degree 2 (quadratic), degree 3 (cubic), and so on up to degree 10.

Create Splines Using Fit Points

When you create splines using fit points, the resulting curve passes through the specified points, and is influenced by the spacing of mathematical *knots* in the curve.

You can choose the spacing of these knots with the *knot parameterization* option, which will result in different curves as shown in the example.



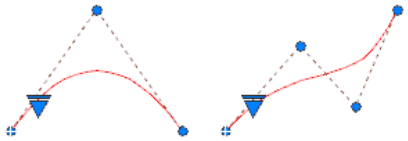
NOTE There is no best choice for knot parameterization for all cases. The chord length parameterization is commonly used, and the square root (centripetal) parameterization often produces better curves depending on the data set.

When the Tolerance value is set to 0, the spline passes directly through the fit points. With larger tolerance values, the spline passes near the fit points. Optionally, you can specify the tangent direction for the spline at each end.

NOTE The fit point method always results in a degree 3 spline.

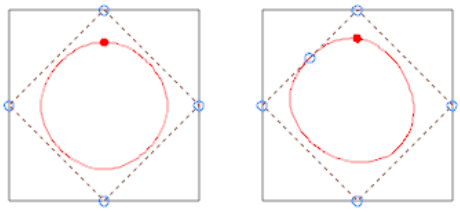
Special Cases

You can create a spline with a parabolic shape by specifying a degree 2 spline created with exactly 3 control vertices as shown on the left. Degree 3 splines created with 4 control vertices have the same shape as Bezier curves of degree 3 as shown on the right.



You can close a spline so that the start point and end point are coincident and tangent. By default, closed splines are mathematically *periodic*, meaning that they have the smoothest (C2) continuity at the point of closure.

In the example, both splines are closed, and the point of closure is marked with a dot. The result of setting the SPLPERIODIC system variable to periodic is shown on the left, while the result of the legacy setting is shown on the right.



NOTE The legacy method for creating B-splines by creating a polyline, and then using the Spline option of the PEDIT command generates only an approximate "spline-fit" polyline.

See also:

- Modify Splines on page 853
- Break and Join Objects on page 843

■ Create Solids and Surfaces from Lines and Curves on page 1030

To draw a spline

1. Click Home tab ➤ Draw panel ➤ Spline. 
2. (Optional) Enter **m** (Method). Then enter either **f** (Fit Points) or **cv** (Control Vertices).
3. Specify the first point of the spline.
4. Specify the next point of the spline. Continue specifying points as needed.
5. Press Enter to end, or enter **c** (Close) to close the spline.

To draw a spline tangent to and connecting two lines or curves

1. Click Modify tab ➤ Blend Curves. 
2. (Optional) Enter **con** (Continuity) and enter a tangency option.
3. Select a line or curve near an endpoint.
4. Select another line or curve near an endpoint.

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.

Quick Reference

Commands

BLEND

Creates a spline in the gap between two selected lines or curves.

CVHIDE

Turns off the display of control vertices for all NURBS surfaces and curves.

CVSHOW

Displays the control vertices for specified NURBS surfaces or curves.

PEDIT

Edits polylines and 3D polygon meshes.

PLINE

Creates a 2D polyline, a single object that is composed of line and arc segments.

SPLINE

Creates a smooth curve that passes through or near a set of fit points, or that is defined by the vertices in a control frame.

SPLINEDIT

Modifies the parameters of a spline or converts a spline-fit polyline to a spline.

System Variables

PLINECONVERTMODE

Specifies the fit method used in converting splines to polylines.

SPLPERIODIC

Controls whether closed splines and NURBS surfaces are generated with periodic properties to maintain the smoothest continuity at the closure point or seam.

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 858
- Create a Solid or Surface by Sweeping on page 1038

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.

Quick Reference

Commands

HELIX

Creates a 2D spiral or 3D spring.

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.

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.

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.

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 (also known as xlines) can be placed anywhere in three-dimensional space. You can specify its orientation in several ways. The

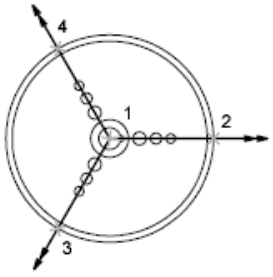
default method for creating the line is the two-point method: you specify two points to define the orientation. The first point, the root, is the conceptual midpoint of the construction line, that is, the point snapped to by the Midpoint object snap.

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.

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.

Quick Reference

Commands

RAY

Creates a line that starts at a point and continues to infinity.

XLINE

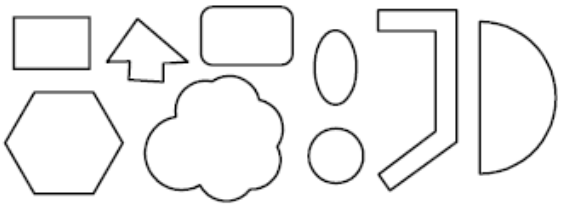
Creates a line of infinite length.

Create and Combine Areas (Regions)

Regions are 2D enclosed areas that have physical properties such as centroids or centers of mass. You can combine existing regions into a single, complex region.

Regions can be used for

- Extracting design information, such as areas and centroids, using MASSPROP
- Applying hatching and shading
- Combining simple objects into more complex ones with Boolean operations.



shapes that can form regions

You can create regions from objects that form closed *loops*. Loops can be combinations of lines, polylines, circles, arcs, ellipses, elliptical arcs, and splines that enclose an area.

You create regions using the REGION command to convert a closed object into a region, and the BOUNDARY command to create a region from an area enclosed by objects. You can combine regions by unifying, subtracting, or intersecting them.

Objects combined using UNION:



selected regions

result

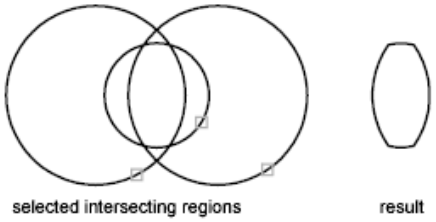
Objects combined using SUBTRACT:



selected regions

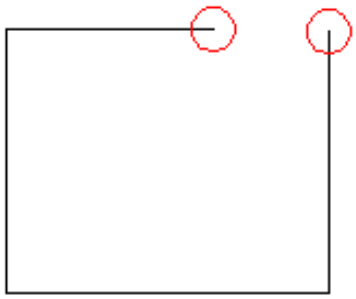
result - a composite region

Objects combined using INTERSECT:



Invalid Boundaries

When a boundary cannot be determined, it might be because the specified internal point is not within a fully enclosed area. With the BOUNDARY command, red circles are displayed around unconnected endpoints of the boundary to identify gaps in the boundary.



The red circles remain displayed even after you exit the command. They are removed when you specify a closed boundary, or by using REDRAW, REGEN, or REGENALL.

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.

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.

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.

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.

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.

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 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 other objects is retained or deleted.

Create Revision Clouds

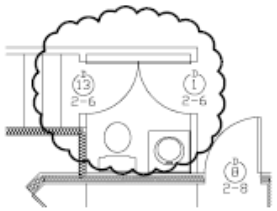
Revision clouds are polylines that consist of sequential arcs. They are used to call attention to parts of a drawing during the review stage.

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.

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.

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.

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.

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 other objects is retained or deleted.

Select and Modify 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.

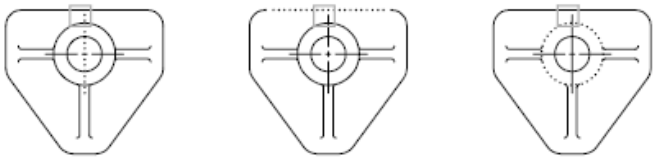
Use the Pickbox Cursor

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 Overlapping or Close Objects

It is sometimes difficult to select objects that overlap or are close together. In the illustration, two lines and a circle lie within the pickbox.



If selection preview is turned on, you can cycle through the objects by rolling over the object on top to highlight it, or you can press and hold Shift and then press the Spacebar. When the required object is highlighted, left-click to select it.

If selection preview is turned off, hold down Shift and Spacebar at a Select Objects prompt while you left-click to cycle through the objects until the one you want is highlighted, and then press Enter . 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 Subobjects on page 1186
- Modify Composite Solids and Surfaces on page 1205

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.

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

PROPERTIES

Controls properties of existing objects.

SELECT

Places selected objects in the Previous selection set.

3DSELECTIONMODE

Controls the selection precedence of both visually and physically 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 for object selection.

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.

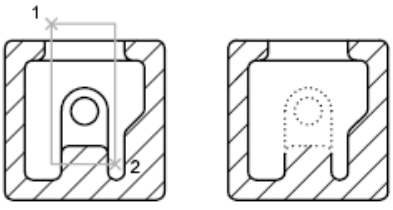
Select Multiple Objects

At the Select Objects prompt, you can select many objects at the same time.

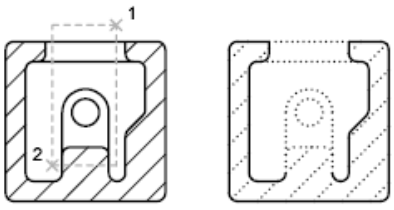
Specify a Rectangular Selection Area

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.



objects selected using window selection box

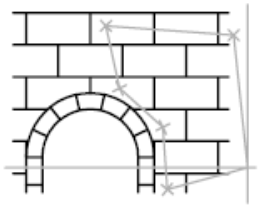


objects selected using crossing selection box

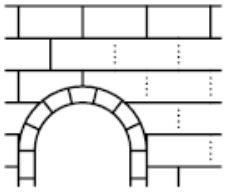
With a window selection, usually the entire object must be contained in the rectangular selection area. However, if an object with a noncontinuous (dashed) linetype is only partially visible in the viewport and all the visible vectors of the linetype can be enclosed within the selection window, the entire 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.



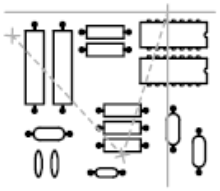
window polygon



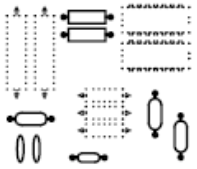
result

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

PROPERTIES

Controls properties of existing objects.

QSELECT

Creates a selection set based on filtering criteria.

SELECT

Places selected objects in the Previous selection set.

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 for object selection.

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.

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.

Quick Reference

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.

LAYLOCKFADECTL

Controls the amount of fading for objects on locked layers.

Select Objects by Properties

Use object properties or object types to include objects in a selection set, or to exclude them.

To quickly define a selection set based on specified filtering criteria, use

- Quick Select (QSELECT) from the Properties palette to filter selection sets by property (such as color) and by object type
- Object Selection Filters dialog box (FILTER) to filter selection sets by property and by object type
- Select Similar (SELECTSIMILAR) to select similar objects of the same type based on specified matching properties
- AutoLISP function SSGET to create a custom selection filter

If you added a feature classification to an object using Autodesk or a third-party application, you can select objects by classification property. With object selection filters, you can name and save filters for future use.

With object selection filters, to filter your selection set based on color, linetype, or lineweight, 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.

By default, objects of the same type are considered similar if they are on the same layer, and, for blocks and other referenced objects, have the same name. Subobjects are only considered at the object level. For example, when a mesh vertex is selected, SELECTSIMILAR selects other mesh objects, not just the mesh vertices.

NOTE If an application such as AutoCAD® Map 3D 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.

See also:

- Customize Object Selection on page 753
- Work with Layers on page 514

To create a selection set using Quick Select

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, specify the color.
7. Under How to Apply, select Include in New Selection Set.
8. Click OK.

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.

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.

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.

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.

To create a selection set of similar objects

1. Enter **selectsimilar**.
2. Enter **se** (settings).
3. In the Select Similar Settings dialog box, select the properties that must match for objects of the same type to be considered similar.

   If no properties are selected, objects of the same type are selected.
4. Click OK.
5. Select object(s) as the selection criteria.
6. Press Enter to select similar objects.

Quick Reference

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.

SELECTSIMILAR

Adds similar objects to the selection set based on selected objects.

PICKADD

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

PICKAUTO

Controls automatic windowing for object selection.

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.

SELECTSIMILARMODE

Controls which properties must match for an object of the same type to be selected with SELECTSIMILAR.

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 with the SELECTIONPREVIEW system variable. 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 performance by setting the HIGHLIGHT system variable to 0. Turning off selection highlighting does *not* affect grips on selected objects.

Set Up Default Selection Methods

The default selection methods are:

- 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.

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.

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.

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

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.

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 for object selection.

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.

See also:

- Work with Blocks on page 915

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, groups 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

CLASSICGROUP

Opens the legacy Object Grouping dialog box.

GROUP

Creates and manages saved sets of objects called groups.

GROUPDISPLAYMODE

Controls the display and grips on groups, when group selection is on.

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.

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.

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.

When group selection is on (PICKSTYLE system variable set to 1 or 3), selecting a member of an exitsting group for inclusion in a new group, selects all members of the former group. To enable individual seleciton of grouped objects turn group selection off (PICKSTYLE set to 0 or 2).

Unnamed groups are not listed in the Object Grouping dialog box unless you select Include Unnamed.

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 an unnamed group

1. Select the objects to associate into a group.
2. Click Home tab ➤ Groups panel ➤ Group. 

   The selected objects are grouped together into an unnamed group, which is assigned a default name such as *A1.

   Unnamed groups are not displayed in the Object Grouping dialog box unless Include Unnamed Groups is selected.

To create a named group

1. Click Home tab ➤ Groups panel ➤ Group. 
2. At the prompt, enter **n** and type a name for the group.
3. Select the objects to group and press Enter.

   The named group is created for the selected objects.

Quick Reference

CLASSICGROUP

Opens the legacy Object Grouping dialog box.

GROUP

Creates and manages saved sets of objects called groups.

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.

By default, groups are selectable; that is, selecting any member of a group selects all the objects in that group. You can then edit the group as a unit. Selecting an object that belongs to multiple groups selects all groups to which that object belongs. Turn off group selection to select grouped objects individually.

TIP Toggle group selection off or on 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). To select groups for editing with grips, use the pointing device to select the group at the Command prompt.

Select a Group

The easiest way to select a group, with group selection on, is to click a grouped object. However, if you need to access a nested group or find and select a specific group, use the Object Grouping dialog box. You also can select groups by name by entering **group** and the group name at any Select Objects prompt.

To control whether all grouped objects can be selected individually

Do one of the following:

- Click Home tab ➤ Groups panel ➤ Group Selection On/Off 
- 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. Click Home tab ➤ Groups panel ➤ Named Groups. 

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.

TIP At the prompt, enter CLASSICGROUP to open the legacy Object Grouping dialog box.

To identify which groups an object belongs to

1 Click Home tab ➤ Groups Panel ➤ Named Groups. 

2 In the Object Grouping dialog box, under Group Identification, click Find Name.

3 In the drawing area select an object.

The Group Member List dialog displays the names of all groups to which the selected object belongs.

4 Click OK to return to the Object Grouping dialog.

Quick Reference

CLASSICGROUP

Opens the legacy Object Grouping dialog box.

GROUP

Creates and manages saved sets of objects called groups.

GROUPDISPLAYMODE

Controls the display and grips on groups, when group selection is on.

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 762.

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

You can specify objects to be added to or removed from a group at any time. You can also rename a group or change it's 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 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, to ungroup the contained objects.

As a result, objects are ungrouped but not otherwise changed.

To delete a group

1. In the drawing area, select a group.
2. Click Home tab ➤ Groups panel ➤ Ungroup. 
   The objects are ungrouped.

To reorder group members

1. Click Home tab ➤ Groups panel ➤ Named Groups. 
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.

To add or remove objects

1. Click Home tab ➤ Groups panel ➤ Group Edit. 
2. In the drawing area, select a group'
3. At the prompt, do one of the following:
   - enter **a** to add objects
   - enter **r** to remove objects
4. Select objects to add to, or remove from, the group.
5. Press Enter.
   The objects are added to, or removed from, the selected group.

To rename a group

1. Click Home tab ➤ Groups panel ➤ Group Edit. 
2. In the drawing area, select a group'

3 At the prompt, enter **rename**.

4 Enter a new name.

5 Press Enter.
The group is renamed.

Quick Reference

CLASSICGROUP

Opens the legacy Object Grouping dialog box.

GROUPEDIT

Adds and removes objects from the selected group, or renames a selected group.

UNGROUP

Disassociates the objects from a group.

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 the UNDO or U commands 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 769.

Cancel a Command

You can cancel a command without completing it by pressing Esc.

To undo the most recent action

- Click Edit menu ➤ Undo.

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.

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.

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.

Quick Reference

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.

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.

Delete Duplicate Objects

You can remove duplicate and overlapping lines, arcs, polylines, and segments of these object types with OVERKILL. Set a tolerance value and specify whether object properties such as layer, color, or plot style are honored or ignored when comparing suspected duplicate objects. OVERKILL also provides a method to consolidate objects.

Clean Up the Display

You can remove stray pixels that are left over from some editing operations from the display area with the REGEN or REGENALL commands.

See also:

- Correct Mistakes on page 766

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.

To delete duplicate objects

1. Click Home tab ➤ Modify panel ➤ Delete Duplicate Objects 
2. At the Select Objects prompt, use a selection method to select objects.
3. Press Enter.
   The Delete Duplicate Objects dialog box is displayed.
4. Optional:
   - Specify a Tolerance value.
   - Select properties to ignore during object comparison.
   - Set additional options to optimize polyline segments, combine objects, or maintain associativity.
5. Click OK.

Duplicate objects are deleted.

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.

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.

To purge an unused linetype

1. Click Application menu ➤ Drawing Utilities ➤ 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.

To remove zero-length geometry and empty text objects

1. Click Application menu ➤ Drawing Utilities ➤ Purge. 

   The Purge dialog box displays.
2. Select Purge zero-length geometry and empty text objects.
3. Click Purge.
4. Click Close.

Quick Reference

CUTCLIP

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

ERASE

Removes objects from a drawing.

OOPS

Restores erased objects.

OVERKILL

Removes duplicate or overlapping lines, arcs, and polylines. Also, combines partially overlapping or contiguous ones.

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.

Cut, Copy, and Paste with the Clipboard

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 programs.

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. The information stored in the Clipboard can then be pasted in other programs.

NOTE If the copied object is embedded, updating the original drawing does not update the copy embedded in the other program.

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, use the EXPLODE command to 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.

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.

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.

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.

Quick Reference

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.

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 modify the size, shape, and location of objects.

See also:

- Work with Custom and Proxy Objects on page 1966
- Modify Existing Dimensions on page 1592

Choose a Method to Modify Objects

Access object editing options using the following methods:

Methods	Descriptions
Command line	Enter a command and then select the objects to modify. Alternatively, select the objects first and then enter a command.
Shortcut menu	Select and right-click an object to display a shortcut menu with relevant editing options.
Double-click	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.)
Grips	Use grips to reshape, move, rotate and manipulate objects: ■ **Grip modes.** Select an object grip to work with the default grip mode—*stretch*—or press Enter or Spacebar to cycle through the additional grip modes—*move*, *rotate*, *scale*, and *mirror*. ■ **Multi-functional grips.** For many objects, you can also *hover* over a grip to access a menu with object-specific, and sometimes grip-specific, editing options.

See also:

- Change Text on page 1483
- Select Objects on page 741
- Modify Existing Dimensions on page 1592

- Display and Change the Properties of Objects on page 509
- Work with Custom and Proxy Objects on page 1966
- "Double Click Actions" in the *Customization Guide*

Quick Reference

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.

DBLCLKEDIT

Controls the double click editing behavior in the drawing area.

DRAGMODE

Controls the way dragged objects are displayed.

GRIPMULTIFUNCTIONAL

Specifies the access methods for multi-functional grip options.

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.

Modify Objects Using Grips

Grips are displayed at strategic points on selected objects.

Use Object Grips

You can reshape, move, or manipulate objects in other ways using different types of grips and grip modes.

Overview

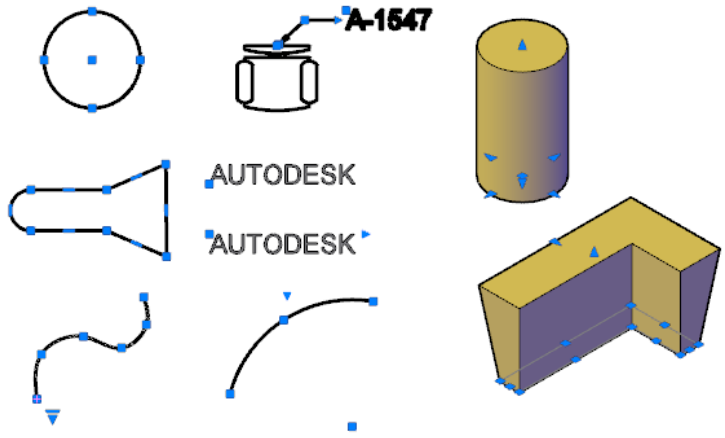
You can use grips in different ways:

- **Use grip modes.** Select an object grip to work with the default grip mode—*stretch*—or press Enter or Spacebar to cycle through the additional grip modes—*move*, *rotate*, *scale*, and *mirror*. You can also right-click a selected grip to see all available options on a shortcut menu.
- **Use multi-functional grips**. For many objects, you can also hover over a grip to access a menu with object-specific, and sometimes grip-specific, editing options. Press Ctrl to cycle through the grip menu options.

Objects with Multi-Functional Grips

The following objects have multi-functional grips that offer object-specific and, in some cases, grip-specific options:

- **2D objects:** Lines, polylines, arcs, elliptical arcs, and splines.
- **Annotation objects:** Dimension objects and multileaders.
- **3D solids:** 3D faces, edges, and vertices.



Important Notes

- Grips are not displayed on objects that are on locked layers.
- When you select multiple objects that share coincident grips, you can edit these objects using grip modes; however, any object- or grip-specific options are not available.

Tips for Stretching with Grips

- When you select more than one grip on an object to stretch it, 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.
- Grips on text, block references, midpoints of lines, centers of circles, and point objects move the object rather than stretching it.
- 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.
- If you select a quadrant grip to stretch a circle or ellipse and then specify a distance at the Command prompt for the new radius—rather than moving the grip—this distance is measured from the center of the circle, not the selected grip.

Limit the Display of Grips to Improve Performance

You can limit the maximum number of objects that display grips. For example, when a drawing contains hatch objects or polylines with many grips, selecting these objects can take a long time. 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.

See also:

- Choose a Method to Modify Objects on page 776
- Modify Polylines on page 846
- Modify Hatches and Fills on page 1389
- Use Dynamic Input on page 619
- Use 3D Subobject Grips on page 1176

- Modify Leaders Using Grips on page 1449
- Modify Dimension Geometry on page 1593
- Work with Dynamic Blocks in Drawings on page 922

To turn on grips

1. At the Command prompt, enter options.
2. In the Options dialog box, Selection tab, select Show Grips.
3. Click OK.

To edit objects using grips

1. Select the objects to edit.
2. Do one or more of the following:
   - Select and move grips to stretch the object.

     NOTE In the case of some object grips, for example, block reference grips, *stretch* will *move* the object rather than stretch it.

   - Press Enter or Spacebar to cycle to the *move*, *rotate*, *scale*, or *mirror* grip modes, or right-click the selected grip to view a shortcut menu with all available grip modes and other options.
   - Hover over a grip to view and access the multi-functional grip menu (if available), and press Ctrl to cycle through the available options.
3. Move the pointing device and click.

TIP To copy the object, press and hold Ctrl until you have clicked to reposition the grip.

To stretch multiple objects using grips

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.
4. Move the pointing device and click.

Quick Reference

OPTIONS

Customizes the program settings.

DYNINFOTIPS

Controls whether tips are displayed for using Shift and Ctrl when editing with grips.

GRIPBLOCK

Controls the display of grips in blocks.

GRIPCOLOR

Controls the color of unselected grips.

GRIPCONTOUR

Controls the color of the grip contour.

GRIPHOT

Controls the color of selected grips.

GRIPHOVER

Controls the fill color of an unselected grip when the cursor pauses over it.

GRIPMULTIFUNCTIONAL

Specifies the access methods for multi-functional grip options.

GRIPOBJLIMIT

Suppresses the display of grips when the selection set includes more than the specified number of objects.

GRIPS

Controls the display of grips on selected objects.

GRIPSIZE

Sets the size of the grip box in pixels.

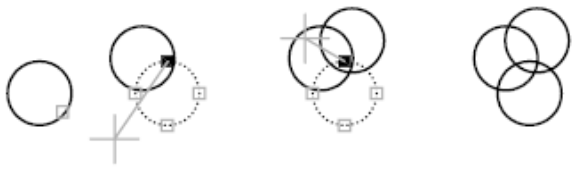
GRIPTIPS

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.

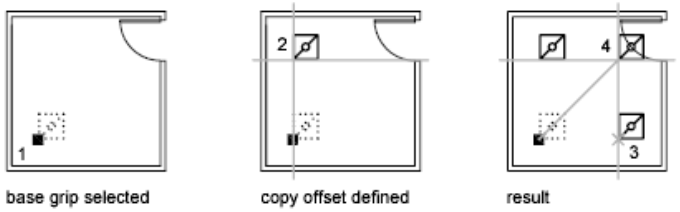


You can also make multiple copies by holding down Ctrl as you select the first point. For example, with the Stretch grip mode, you can stretch an object, such as a line, and then copy it to any point in the drawing area. Multiple copies continue being made until you turn off grips.

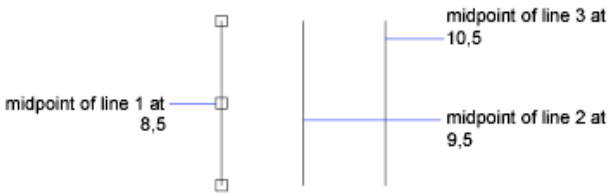
NOTE When you use grips to make multiple copies of an annotative object that contains multiple scale representations, only the current scale representation is copied.

Define an Offset Snap or a Rotation Snap

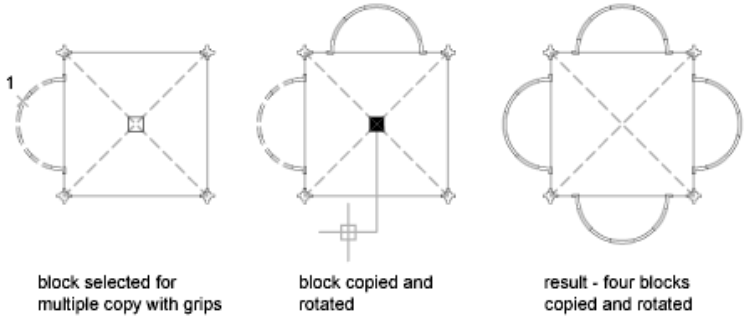
You can place multiple copies at regularly spaced intervals with an offset snap. The offset snap is defined by the distance between an object and the next copy. In the lighting layout below, the first copy of the light fixture symbol is placed at an offset of two units. All subsequent copies are then 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

OPTIONS

Customizes the program settings.

GRIPBLOCK

Controls the display of grips in blocks.

GRIPCOLOR

Controls the color of unselected grips.

GRIPCONTOUR

Controls the color of the grip contour.

GRIPHOT

Controls the color of selected grips.

GRIPS

Controls the display of grips on selected objects.

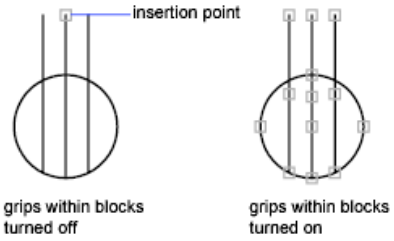
GRIPSIZE

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:

- Use Object Grips on page 778
- Specify Grips for Dynamic Blocks on page 1011

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.

Quick Reference

OPTIONS

Customizes the program settings.

GRIPBLOCK

Controls the display of grips in blocks.

GRIPCOLOR

Controls the color of unselected grips.

GRIPCONTOUR

Controls the color of the grip contour.

GRIPHOT

Controls the color of selected grips.

GRIPS

Controls the display of grips on selected objects.

GRIPSIZE

Sets the size of the grip box in pixels.

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.

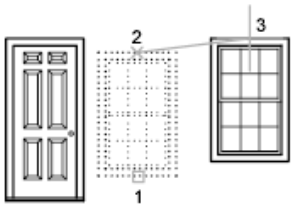
Move Objects

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

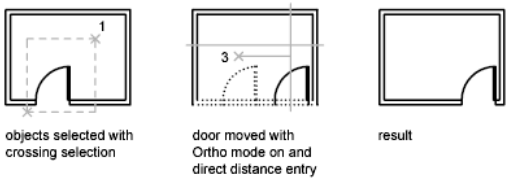
Move an object using the distance and direction specified by a base point followed by a second point. In this example, you move the block representing a window. Select the object to be moved (1). Specify the base point for the move (2) followed by a second point (3). The object is moved the distance and direction of point 2 to point 3.



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.



The result is that only the endpoints that lie within the crossing selection move.

Drag, Grip-Edit, or Nudge Objects

Move selected objects quickly by dragging , grip-editing, or nudging.

Drag objects within a drawing, or between open drawings and other applications. Dragging with the right button on the pointing device allows you to specify whether to move, copy, or create a block from the dragged objects. Dragging disregards all snap settings.

To nudge selected objects in orthogonal increments, press Ctrl + arrow keys. Snap mode affects the distance and direction in which the objects are nudged.

- **Nudge objects with Snap mode turned off:** Objects move two pixels at a time; movement is relative and orthogonal to the screen, regardless of the view direction or the UCS orientation.

- **Nudge objects with Snap mode turned on:** Objects are moved in increments specified by the current snap spacing; movement is orthogonal to the X and Y axes of the current UCS and relative to the view direction.

See also:

- Modify Objects Using Grips on page 777
- Embed OLE Objects in Drawings on page 1830

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.

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.

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.

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.

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.

System Variable

SNAPUNIT

Sets the snap spacing for the current viewport.

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.

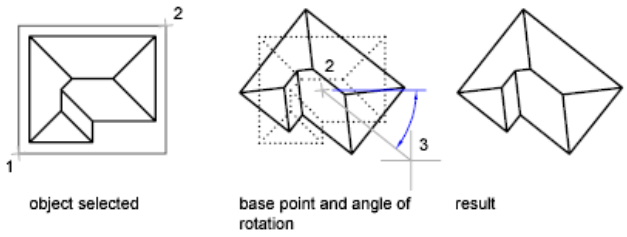
Rotate an Object by a Specified Angle

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 base angle direction 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.

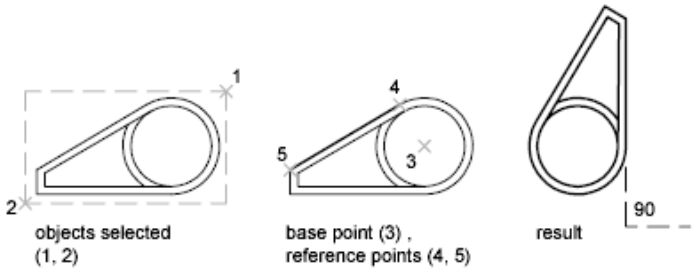
For example, you can rotate the plan view of a house by selecting the objects (1), specifying a base point (2), and specifying an angle of rotation by dragging to another point (3).



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 463

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.

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.

To rotate a 3D object around an axis

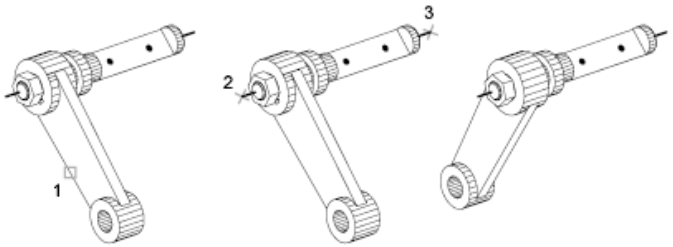
1 Click Modify menu ➤ 3D Operations ➤ Rotate 3D. 

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 "Use World and User Coordinate Systems in 3D" on page 15).

4 Specify the angle of rotation.



Quick Reference

Commands

ROTATE

Rotates objects around a base point.

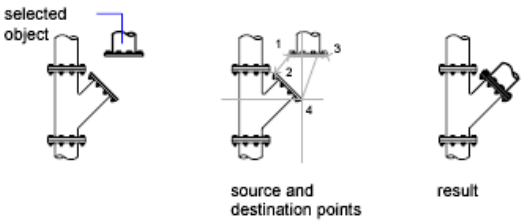
ROTATE3D

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.

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.

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, Array, Offset, or Mirror Objects

You can create duplicates of objects in your drawing that are either identical or similar to selected objects.

Copy Objects

See also:

- Modify Objects Using Grips on page 777

- Enter Direct Distances on page 655
- Embed OLE Objects in Drawings on page 1830
- Array Objects on page 799

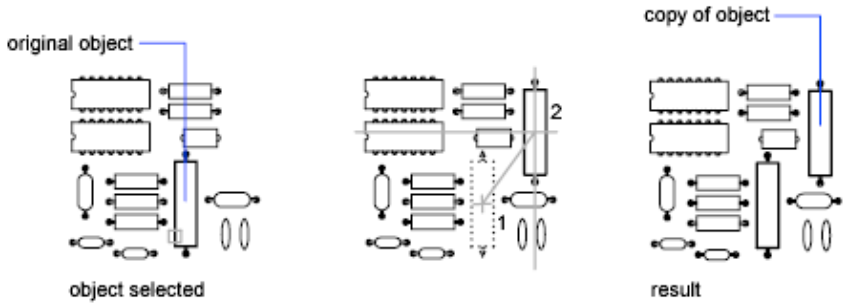
You can create duplicates of objects at a specified distance and direction from the originals.

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.

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. 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.

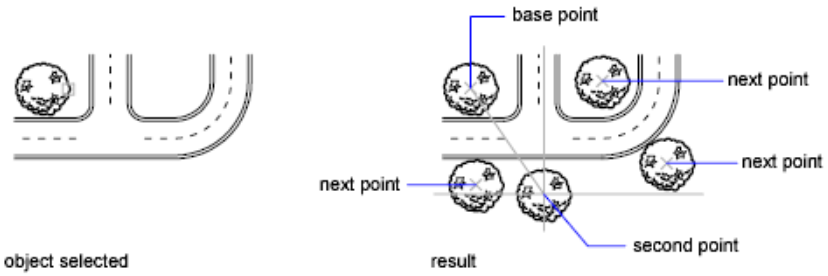
To copy objects a specified distance, you can also use direct distance entry with Ortho mode and polar tracking.

Create Multiple Copies

With COPY, you can create multiple copies from the specified selection set and base point.

The options include the following:

- Create copies at specified locations or displacements
- Automatically space a specified number of copies in a linear pattern.



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.

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.

To create an object based on a selected object

1. Select the object on which to base the new object.
2. Right-click and select Add Selected.
3. Follow the prompts to create an object similar to the selected object.

Quick Reference

Commands

ADDSELECTED

Creates a new object based on the object type and general properties of a selected object.

COPY

Copies objects a specified distance in a specified direction.

NCOPY

Copies objects that are contained in an xref, block, or DGN underlay.

System Variables

COPYMODE

Controls whether the COPY command repeats automatically.

Array Objects

Create multiple copies of objects that are evenly distributed in a rectangular or circular pattern, or along a specified path.

Overview of Arrays

Create copies of objects arranged in a pattern called an array.

There are three types of arrays:

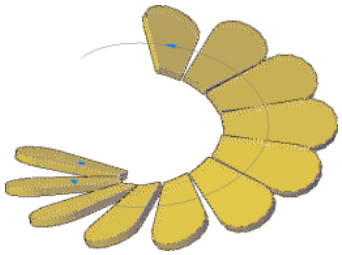
- Rectangular
- Path
- Polar



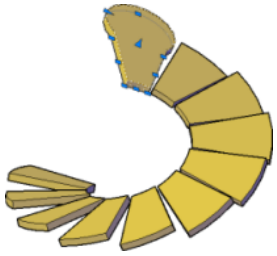
Control Array Associativity

Associativity allows you to quickly propagate changes throughout an array by maintaining relationships between items. Arrays can be associative or non-associative.

- **Associative**. Items are contained in a single *array object*, similar to a block. Edit the array object properties, such as the spacing or number of items. Override item properties or replace an item's source objects. Edit an item's source objects to change all items that reference those source objects.



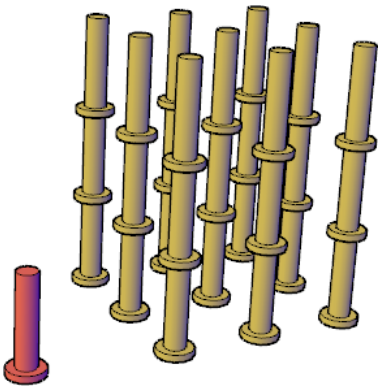
- **Non-associative.** Items in the array are created as independent objects. Changes to one item do not affect the other items.



Create Rectangular Arrays

In rectangular arrays, items are distributed into any combination of rows, columns, and levels.

A dynamic preview allows you to quickly derive the number and spacing of rows and columns. Add levels to make a 3D array.



The number of rows and columns and the spacing between the rows and columns in the array increase or decrease as you move your cursor. By default, the array has a level of 1.

You can rotate the array around the base point in the XY plane. At creation, the row and column axes are orthogonal to each other; for associative arrays, you can later edit the axis angles.

To create a rectangular array

1. Click Home tab ➤ Modify panel ➤ Rectangular Array. 
2. Select the objects to array and press Enter.
3. Specify a point for the opposite corner of the grid to set the number of rows and columns.

   A preview grid displays as you define the array.
4. Specify a point for the opposite corner of the grid to set the row and column spacing.
5. Press Enter.

Quick Reference

Commands

3DARRAY

Maintains legacy behavior for creating nonassociative, 3D rectangular or polar arrays.

ARRAY

Creates copies of objects arranged in a 2D or 3D pattern.

ARRAYCLOSE

Saves or discards changes made to an array's source objects and exits the array editing state.

ARRAYEDIT

Edits associative array objects and their source objects.

ARRAYPATH

Evenly distributes object copies along a path or a portion of a path.

ARRAYPOLAR

Evenly distributes object copies in a circular pattern around a center point or axis of rotation.

ARRAYRECT

Distributes object copies into any combination of rows, columns, and levels.

DSETTINGS

Sets grid and snap, polar and object snap tracking, object snap modes, Dynamic Input, and Quick Properties.

UCS

Sets the origin and orientation of the current UCS.

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.

ARRAYEDITSTATE

Indicates whether the drawing is in the array editing state, which is activated while editing an associative array's source objects.

ARRAYTYPE

Specifies the default array type.

DELOBJ

Controls whether geometry used to create other objects is retained or deleted.

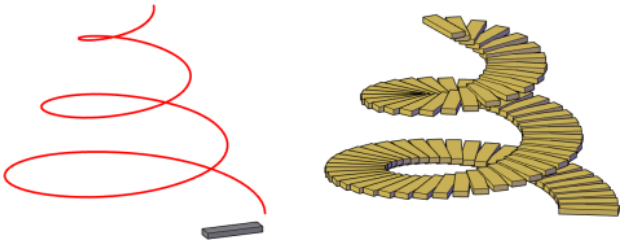
SNAPANG

Sets the snap and grid rotation angle for the current viewport relative to the current UCS.

Create Path Arrays

In path arrays, items are evenly distributed along a path or a portion of a path.

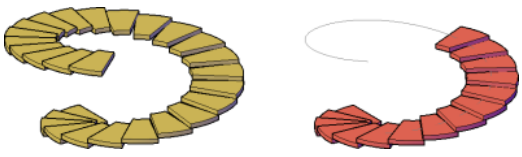
A path can be a line, polyline, 3D polyline, spline, helix, arc, circle, or ellipse.



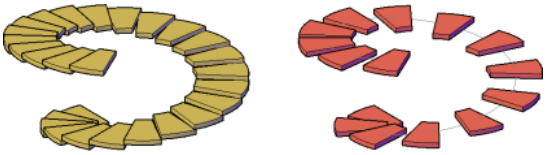
Control Item Distribution

The distribution of items along the path can be measured or divided.

- **Measure.** The array follows the path when it is edited but the number of objects and spacing do not change. If the path is edited and becomes too short to display all objects, the count is automatically adjusted.



- **Divide.** The number of objects and the length of the path determine the spacing of the objects in the array. The objects are always spaced evenly along the entire length of the path. When the array is associative, the spacing between the objects automatically adjusts as the length of the path changes after it is created.



To create a path array

1. Click Home tab ➤ Modify panel ➤ Path Array. 
2. Select the objects to array and press Enter.
3. Select the path curve.

   A preview displays as you define the array.
4. (Optional) Enter **o** (Orientation) and specify a base point, or press Enter to use the end of the selected path as the base point. And then specify one of the following methods:
   - Direction to align with the starting direction of the path.
   - **Normal.** Objects are aligned to be normal to the starting direction of the path.
5. Do one of the following:
   - Specify the distance between items.
   - Enter **d** (Divide) to evenly distribute items along the entire length of the path.
   - Enter **t** (Total) and specify the total distance between the first and last items.
   - Enter e (Expression) and define an expression.
6. Press Enter.

Quick Reference

Commands

3DARRAY

Maintains legacy behavior for creating nonassociative, 3D rectangular or polar arrays.

ARRAY

Creates copies of objects arranged in a 2D or 3D pattern.

ARRAYCLOSE

Saves or discards changes made to an array's source objects and exits the array editing state.

ARRAYEDIT

Edits associative array objects and their source objects.

ARRAYPATH

Evenly distributes object copies along a path or a portion of a path.

ARRAYPOLAR

Evenly distributes object copies in a circular pattern around a center point or axis of rotation.

ARRAYRECT

Distributes object copies into any combination of rows, columns, and levels.

DSETTINGS

Sets grid and snap, polar and object snap tracking, object snap modes, Dynamic Input, and Quick Properties.

UCS

Sets the origin and orientation of the current UCS.

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.

ARRAYEDITSTATE

Indicates whether the drawing is in the array editing state, which is activated while editing an associative array's source objects.

ARRAYTYPE

Specifies the default array type.

DELOBJ

Controls whether geometry used to create other objects is retained or deleted.

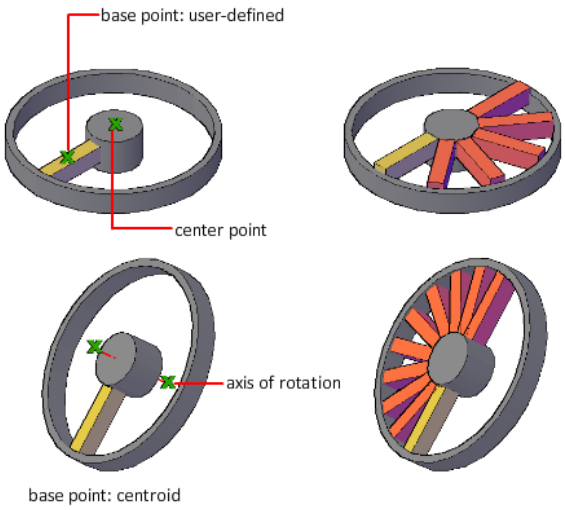
SNAPANG

Sets the snap and grid rotation angle for the current viewport relative to the current UCS.

Create Polar Arrays

In polar arrays, items are evenly distributed in a circular motion around a specified center point or axis of rotation.

When you create a polar array using the center point, the axis of rotation is the Z axis of the current UCS. You can redefine the axis of rotation by specifying two points.



The direction in which the array is drawn depends on whether you enter a positive or negative value for the fill angle. For associative arrays, you can change the direction in the Properties palette.

To create a polar array

1. Click Home tab ➤ Modify panel ➤ Polar Array. 
2. Select the objects to array.
3. Do one of the following:
   - Specify a center point.
   - Specify a base point.
   - Enter **a** (Axis of rotation) and specify two points to define a custom axis of rotation.

   A preview displays as you define the array.
4. Specify the number of items.
5. Specify the angle to fill.
6. Press Enter.
7. Press Enter.

You can also create the array by defining the angle between items.

Quick Reference

Commands

3DARRAY

Maintains legacy behavior for creating nonassociative, 3D rectangular or polar arrays.

ARRAY

Creates copies of objects arranged in a 2D or 3D pattern.

ARRAYCLOSE

Saves or discards changes made to an array's source objects and exits the array editing state.

ARRAYEDIT

Edits associative array objects and their source objects.

ARRAYPATH

Evenly distributes object copies along a path or a portion of a path.

ARRAYPOLAR

Evenly distributes object copies in a circular pattern around a center point or axis of rotation.

ARRAYRECT

Distributes object copies into any combination of rows, columns, and levels.

DSETTINGS

Sets grid and snap, polar and object snap tracking, object snap modes, Dynamic Input, and Quick Properties.

UCS

Sets the origin and orientation of the current UCS.

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.

ARRAYEDITSTATE

Indicates whether the drawing is in the array editing state, which is activated while editing an associative array's source objects.

ARRAYTYPE

Specifies the default array type.

DELOBJ

Controls whether geometry used to create other objects is retained or deleted.

SNAPANG

Sets the snap and grid rotation angle for the current viewport relative to the current UCS.

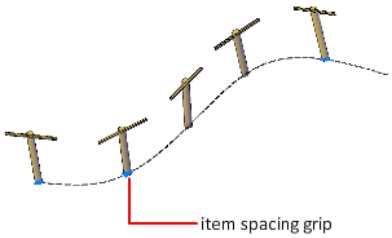
Edit Associative Arrays

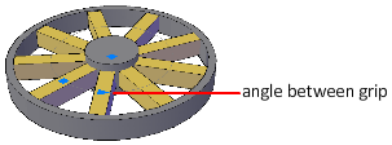
Modify associative arrays by editing the array properties, applying item overrides, replacing selected items, or editing source objects.

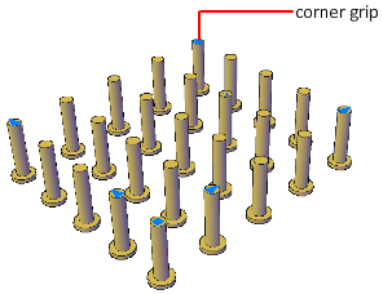
Edit Array Properties

To edit the array properties, use ARRAYEDIT, the Properties palette, or grips.

The type of grips and dynamic menu options displayed depend on the type of array.

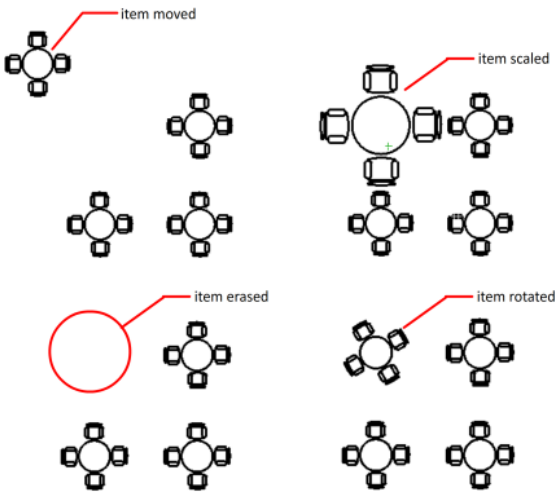






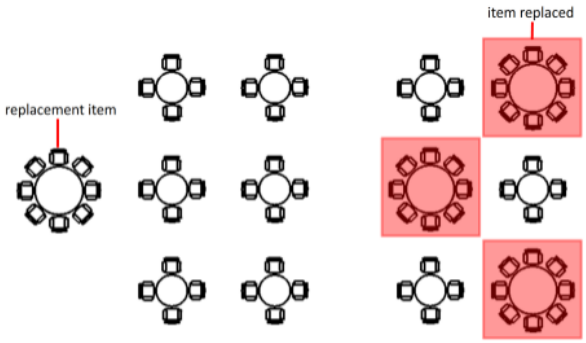
Apply Item Overrides

Ctrl+click items in the array to erase, move, rotate, or scale the selected items without affecting the rest of the array. Reset the array to remove all item overrides.



Replace Items

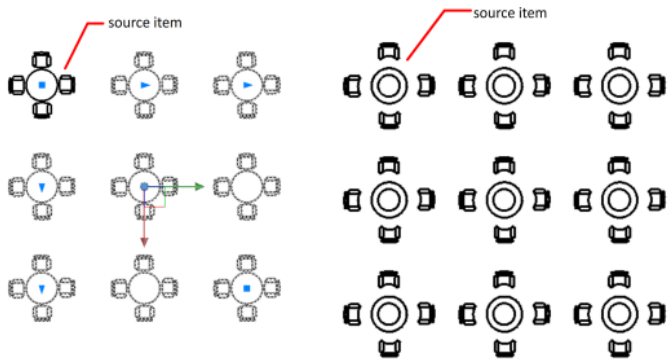
Replace selected items with other objects. Any item overrides are maintained. You can also replace all items that reference the original source objects, rather than selecting individual items.



Edit Source Objects

To edit an item's source objects, activate an editing state for a selected item. All changes (including the creation of new objects) are instantly applied to

all items referencing the same set of source objects. Save or discard your changes to exit the editing state.



To modify the number of items in a rectangular array

1. Select the array.
2. Hover over the Corner grip to display the grip menu.

   The *Corner grip* is displayed on the first item in the last column of the rectangular array.
3. Click the Row and Column Count option.
4. Move the cursor to increase or decrease the number of rows and columns and then click.

To adjust the item spacing between objects in a path array

1. Select the array.
2. In the Properties palette, set the Method property to Measure.
3. Click the Item Spacing grip.

   The *Item Spacing grip* is displayed on the second item in the first row of the path array. This grip is displayed only when there are three or more items in the array.
4. Move the cursor along the path to increase or decrease the distance between the items and then click.

To modify the angle between the items in a polar array

1. Select the array.

2. Click the Angle Between grip.

   The *Angle Between grip* is displayed on the second item in the first row of the polar array. This grip is displayed only when there are three or more items in the array.

3. Move the cursor to increase or decrease the angle between the items and then click.

Quick Reference

Commands

3DARRAY

Maintains legacy behavior for creating nonassociative, 3D rectangular or polar arrays.

ARRAY

Creates copies of objects arranged in a 2D or 3D pattern.

ARRAYCLOSE

Saves or discards changes made to an array's source objects and exits the array editing state.

ARRAYEDIT

Edits associative array objects and their source objects.

ARRAYPATH

Evenly distributes object copies along a path or a portion of a path.

ARRAYPOLAR

Evenly distributes object copies in a circular pattern around a center point or axis of rotation.

ARRAYRECT

Distributes object copies into any combination of rows, columns, and levels.

DSETTINGS

Sets grid and snap, polar and object snap tracking, object snap modes, Dynamic Input, and Quick Properties.

UCS

Sets the origin and orientation of the current UCS.

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.

ARRAYEDITSTATE

Indicates whether the drawing is in the array editing state, which is activated while editing an associative array's source objects.

ARRAYTYPE

Specifies the default array type.

DELOBJ

Controls whether geometry used to create other objects is retained or deleted.

SNAPANG

Sets the snap and grid rotation angle for the current viewport relative to the current UCS.

Limit the Size of Arrays

The number of array elements that can be generated by one ARRAY command is limited to approximately 100,000.

This limit is controlled by the MaxArray setting in the registry. If you specify a large number of rows and columns for an array, it may take a long time to create the copies.

You can change the limit by setting the MaxArray system registry variable using (**setenv "MaxArray" "n"**) where *n* is a number from 100 through 10000000 (ten million).

The MaxArray system registry variable is validated when using the ARRAYEDIT command. The Array Limit Object task dialog is displayed when the total count of the modified array exceeds the current MaxArray value.

NOTE When changing the value of MaxArray, you must enter MaxArray with the capitalization shown.

Quick Reference

Commands

3DARRAY

Maintains legacy behavior for creating nonassociative, 3D rectangular or polar arrays.

ARRAY

Creates copies of objects arranged in a 2D or 3D pattern.

ARRAYCLOSE

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ARRAYRECT

Distributes object copies into any combination of rows, columns, and levels.

DSETTINGS

Sets grid and snap, polar and object snap tracking, object snap modes, Dynamic Input, and Quick Properties.

UCS

Sets the origin and orientation of the current UCS.

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.

ARRAYEDITSTATE

Indicates whether the drawing is in the array editing state, which is activated while editing an associative array's source objects.

ARRAYTYPE

Specifies the default array type.

DELOBJ

Controls whether geometry used to create other objects is retained or deleted.

SNAPANG

Sets the snap and grid rotation angle for the current viewport relative to the current UCS.

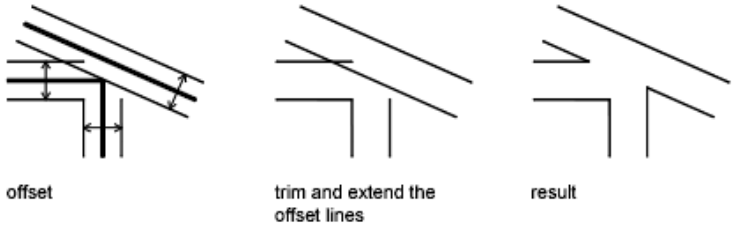
Offset an Object

Offset an object to create a new object whose shape is parallel to the original object.

For example, if you offset a circle or an arc, a larger or smaller circle or arc is created, depending on which side you specify for the offset. If you offset a polyline, the result is a polyline that parallels the original.



TIP An effective drawing technique is to offset objects and then trim or extend their ends.

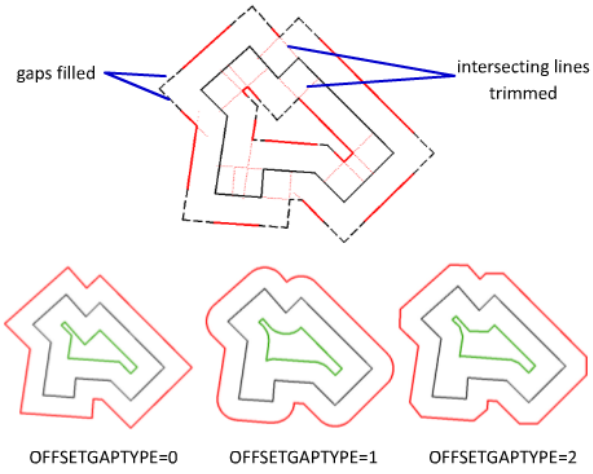


Use OFFSET to offset the following object types:

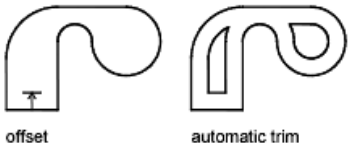
- Lines
- Arcs
- Circles
- Ellipses and elliptical arcs (resulting in an oval-shaped spline)
- 2D polylines
- Construction lines (xlines) and rays
- Splines

Special Cases for Offset Polylines and Splines

2D polylines are offset as individual line segments, resulting in either intersections or gaps between segments. To complete the offset, intersecting lines are trimmed and gaps are filled.



Splines are trimmed automatically when the offset distance is larger than can otherwise be accommodated.



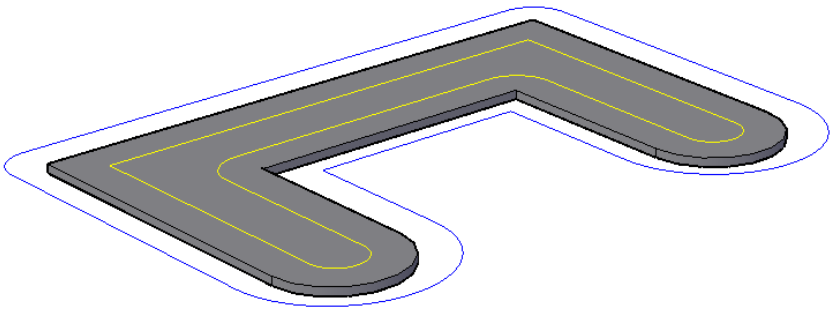
Offset the Edges of Faces on Solids and Surfaces

The OFFSETEDGE command creates a closed polyline or a spline from the edges of a planar face on a 3D solid or surface. Splines result when one or more edge segments cannot be represented as lines, arcs, or a circle. For example

- In the left illustration, the edge of the top surface of the solid was offset, resulting in the closed, yellow polyline shown.
- In the middle illustration, the polyline was extruded.
- In the right illustration, the polyline was extruded and then subtracted.



With the Corner option, you can specify sharp corners or rounded corners for the polyline. External and internal corners create rounded corners differently depending whether the corners are concave or convex—the radius of the arcs maintain the specified offset distance.



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 to indicate whether the object is to be offset inside or outside of the original object.
5. Select another object to offset, or press Enter to end the command.

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 a point through which the offset object is to pass.
5. Select another object to offset, or press Enter to end the command.

To create a polyline or a spline that is offset from the edges of a face

1. Click Solid tab ➤ Solid Editing panel ➤ Offset Edge. 
2. Select a planar face on a 3D solid or surface.
3. Specify a point location inside or outside the edges of the face, or enter an option to specify an offset distance or a corner style.
4. Select another face to offset, or press Enter to end the command.

Quick Reference

Commands

OFFSET

Creates concentric circles, parallel lines, and parallel curves.

OFFSETEDGE

Creates a closed polyline or spline object that is offset at a specified distance from the edges of a selected planar face on a 3D solid or surface.

System Variables

OFFSETDIST

Sets the default offset distance.

OFFSETGAPTYPE

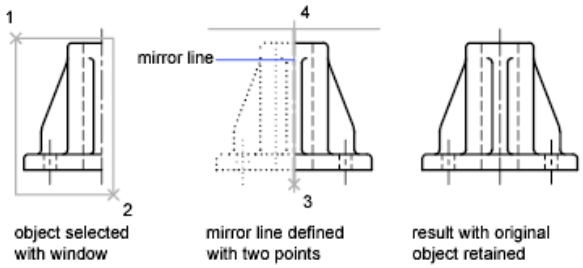
Controls how potential gaps between segments are treated when polylines are offset.

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, hatches, 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.



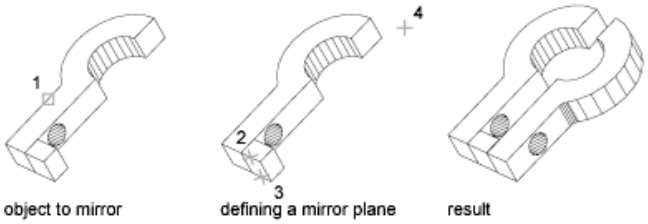
MIRRTEXT affects text that is created with the TEXT, ATTDEF, or MTEXT commands; attribute definitions; and variable attributes. Text and constant attributes that are part of an inserted block are reversed when the block is mirrored regardless of the value of MIRRTEXT.

MIRRHATCH affects hatch objects created with the GRADIENT or HATCH commands. Use the MIRRHATCH system variable control whether hatch pattern direction is mirrored or retained.

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.

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.

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

MIRRHATCH

Controls how MIRROR reflects hatch patterns.

MIRRTEXT

Controls how MIRROR 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.

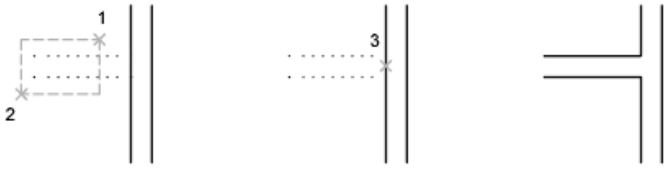
If you do not specify a boundary and press Enter at the Select Objects prompt, all displayed objects become potential boundaries.

NOTE To select cutting edges or boundary edges that include blocks, you can use only the single selection, Crossing, Fence, and Select All options.

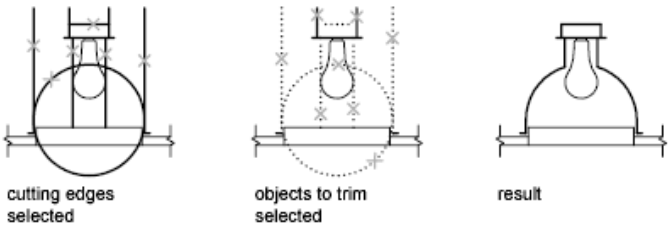
Trim Objects

You can trim objects so that they end precisely at boundary edges defined by other objects.

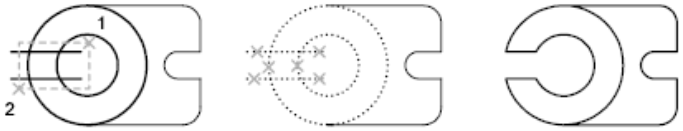
For example, you can clean up the intersection of two walls smoothly by trimming.



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.

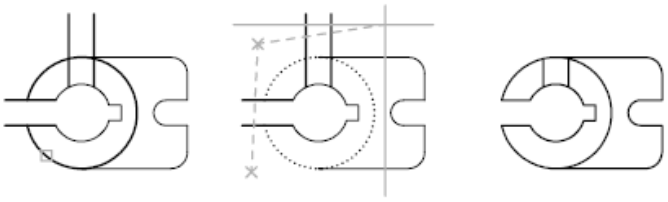


When you trim several objects, the different selection methods can help you choose the current cutting edges and objects to trim. In the following example, the cutting edges are selected using crossing selection.



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.

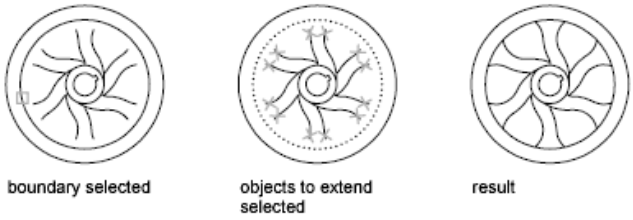


all objects are selected by pressing ENTER | objects to trim selected | result

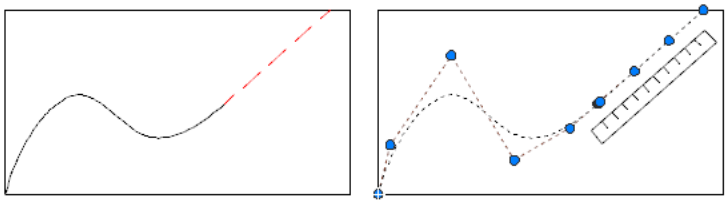
NOTE You can extend objects without exiting the TRIM command. Press and hold Shift while selecting the objects to be extended.

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.



Extending a spline preserves the shape of the original portion of the spline, but the extended portion is linear and tangent to the end of the original spline.

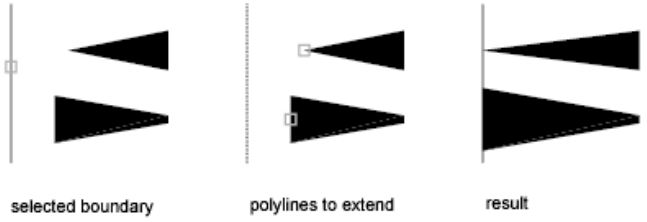


NOTE You can trim objects without exiting the EXTEND command. Press and hold Shift while selecting the objects to be trimmed.

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 843

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.

To trim an object

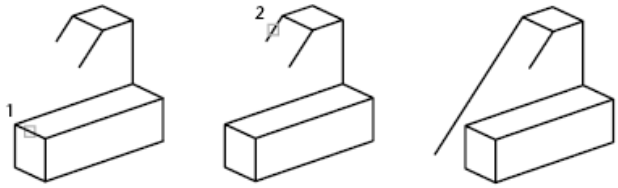


1. Click Home tab ➤ Modify panel ➤ Trim.
2. Select the objects to serve as cutting edges.

   To select all displayed objects as potential cutting edges, press Enter without selecting any objects.
3. Select the objects to trim.

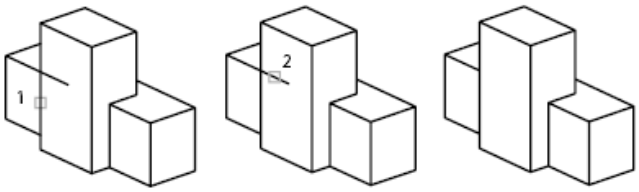
To extend objects in 3D wireframe models

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).



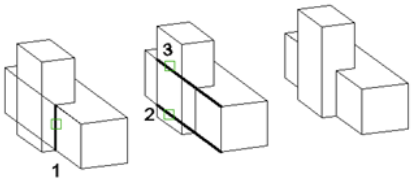
To trim in 3D using the current view plane

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).



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).



Quick Reference

Commands

BREAK

Breaks the selected object between two points.

EXTEND

Extends objects to meet the edges of other objects.

JOIN

Joins the endpoints of linear and curved objects to create a single 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

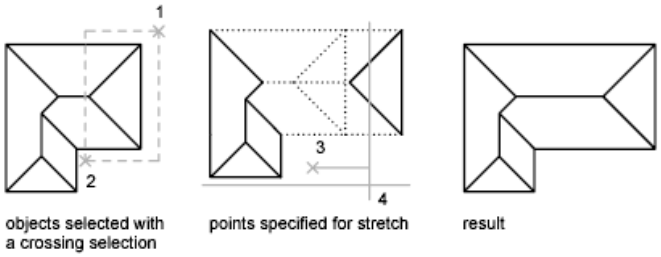
- Drag an object endpoint dynamically
- Specify a new length or angle as a percentage of the total length or angle
- Specify an incremental length or angle measured from an endpoint
- Specify the object's total absolute length or included angle

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.

You stretch an object by specifying a base point and then a point of displacement.



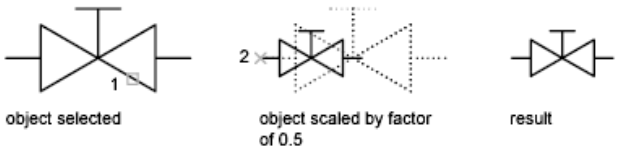
Use object snaps, grid snaps, and relative coordinate entry to stretch with precision.

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 annotative on page 2289 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 843

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.

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.

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.

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.

Quick Reference

Commands

JOIN

Joins the endpoints of linear and curved objects to create a single 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

Modifies the parameters of a spline or converts a spline-fit polyline to a spline.

STRETCH

Stretches objects crossed by a selection window or polygon.

System Variables

PLINECONVERTMODE

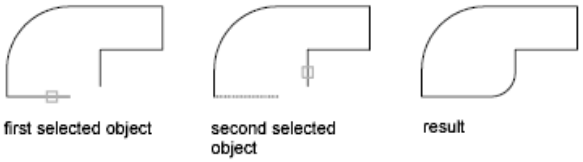
Specifies the fit method used in converting splines to polylines.

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. Also, you can use the Multiple option to fillet more than one set of objects without leaving the 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.

Instead of an arc, you can use BLEND to create a spline that connects two lines or curves, and is tangent to them.

Set the Fillet Radius

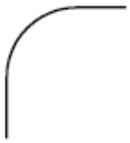
The fillet radius is the radius of the arc that connects filleted objects. Changing the fillet radius affects subsequent fillets. If you set the fillet radius to 0, filleted objects are trimmed or extended until they intersect, but no arc is created.



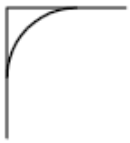
You can press and hold Shift while selecting the objects to override the current fillet radius with a value of 0.

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



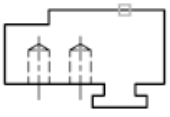
fillet location points selected



result

Fillet Line and Polyline Combinations

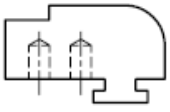
To fillet lines with polylines, each line or its extension must intersect one of the polyline line segments. If the Trim option is on, the filleted objects and the fillet arc join to form a single new polyline.



polyline selected



line selected

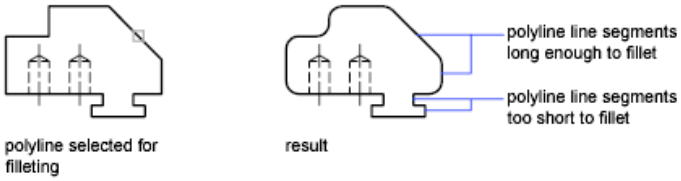


result

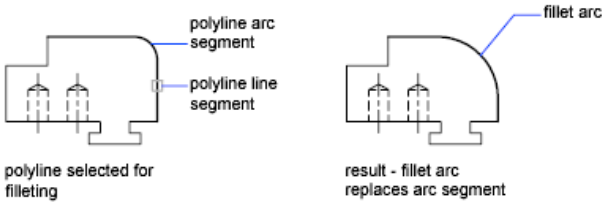
Fillet an Entire Polyline

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 linear segments in a polyline are separated by an arc segment between them, FILLET removes the arc segment and replaces it with a new arc segment of the current fillet radius.

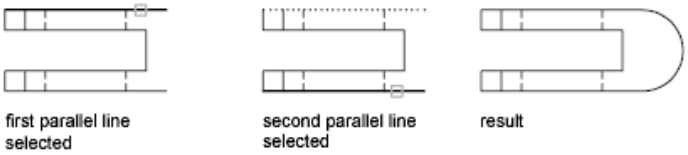


If you set the fillet radius to 0, no fillet arcs are inserted. If two linear polyline segments are separated by one arc segment, FILLET removes that arc and extends the linear segments until they intersect.

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.



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.

To fillet two line segments



1. Click Home tab ➤ Modify panel ➤ Fillet.
2. Select the first line.
3. Select the second line.

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.

To fillet an entire polyline



1. Click Home tab ➤ Modify panel ➤ Fillet.
2. Enter **p** (Polyline).
3. Select the polyline.

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.

Quick Reference

Commands

FILLET

Rounds and fillets the edges of objects.

FILLETEDGE

Rounds and fillets the edges of solid objects.

System Variables

FILLETRAD

Stores the current fillet radius for 2D objects.

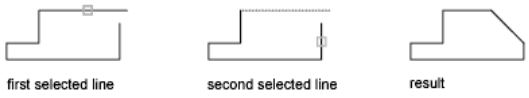
TRIMMODE

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.

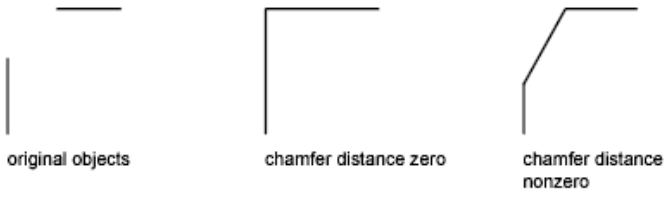
NOTE Chamfering a hatch boundary that was defined from line segments removes hatch associativity. If the hatch boundary was defined from a polyline, associativity is maintained.

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.

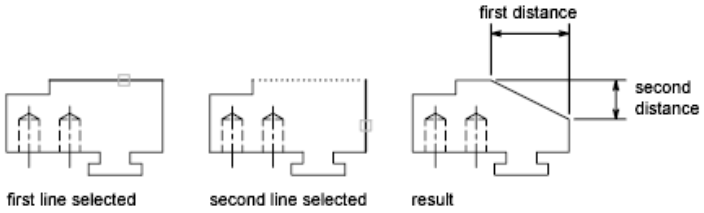
Use the Multiple option to chamfer more than one set of objects without leaving the command.

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 press and hold 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.



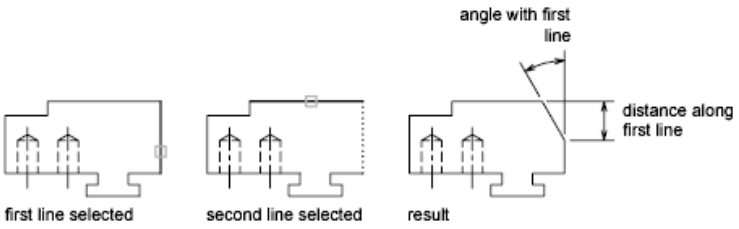
Trim and Extend Chamfered Objects

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

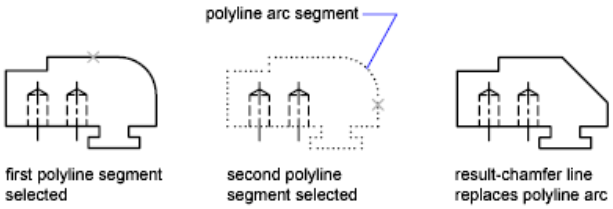
You can chamfer two objects by specifying where on the first selected object the chamfer line starts, and then the angle the chamfer line forms with this object.

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

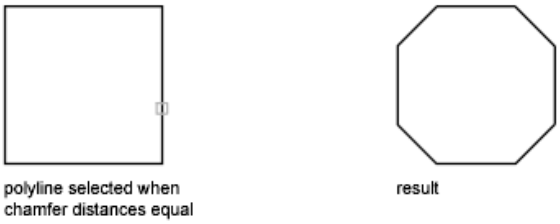
If the two objects you select for chamfering are segments of a polyline, they must be adjacent or separated by no more than one arc segment. If they are separated by an arc segment, as shown in the illustration, chamfering deletes the arc and replaces it with a chamfer line.



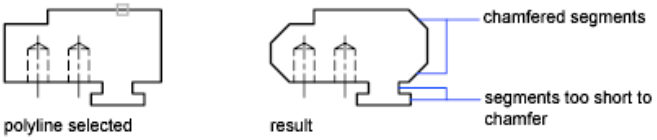
Chamfer an Entire Polyline

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.

To chamfer two nonparallel line segments

1. Click Home tab ➤ Modify panel ➤ Chamfer. 
2. Select the first line.
3. Select the second line.

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.

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.

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.

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.

Quick Reference

Commands

CHAMFER

Bevels the edges of objects.

CHAMFEREDGE

Bevels the edges of 3D solids and surfaces.

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

PREVIEWCREATIONTRANSPARENCY

TRIMMODE

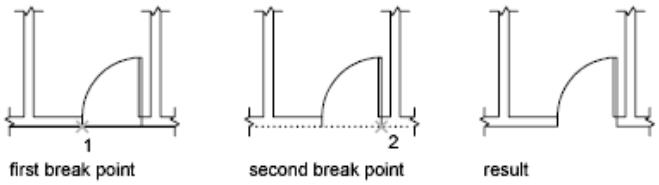
Controls whether selected edges for chamfers and fillets are trimmed.

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 create single object or multiple objects.

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.

You can create breaks in most geometric objects except blocks, dimensions, multilines, and regions. As an alternative, use EXPLODE on these types of objects, and create breaks in the dissociated geometry.

Join Objects

Use JOIN to combine lines, arcs, elliptical arcs, polylines, 3D polylines, helixes, and splines by their endpoints into a single object.

The result of the join operation varies depending on the objects selected. Typical applications include

- Replacing two collinear lines with a single line.
- Closing the gap in a line that resulted from a BREAK.
- Completing an arc into a circle or an elliptical arc into an ellipse. To access the Close option, select a single arc or elliptical arc.

- Combining several long polylines in a topographic map.
- Joining two splines, leaving a kink between them.

In general cases, joining objects that touch end-to-end, but that are not in the same plane result in 3D polylines and splines.

NOTE You can also use the Join option of the PEDIT command to combine a series of lines, arcs, and polylines into a single polyline

See also:

- Modify Polylines on page 846
- Modify Splines on page 853
- Modify Multilines on page 860

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.

To join objects

1. Click Home tab ➤ Modify panel ➤ Join. 
2. Select a source object or select multiple objects to join together.
3. Valid objects include lines, arcs, elliptical arcs, polylines, 3D polylines, and splines.

Quick Reference

Commands

BREAK

Breaks the selected object between two points.

JOIN

Joins the endpoints of linear and curved objects to create a single object.

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.

NOTE Blocks inserted with MINSERT (multiple insert) result in an *minsert block* object, and cannot be exploded directly. You can convert the minsert block object into a block object with the FLATTEN Express Tool.

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.

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 Polylines

Change the shape and display of polyline objects with polyline editing options. You can also join separate polylines.

You can modify polylines using PEDIT, the Properties palette, or grips.

- 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 polyline's linetype by reversing its direction

Modify Polylines with Grips

NOTE For general information about working with grips, see Modify Objects Using Grips on page 777.

Polyline grips offer some grip-specific options, depending on

- The grip's location (vertex or midpoint)
- The segment type (line or arc)
- The type of Polyline (standard, curve-fit, or spline-fit)

Polyline grip menu options

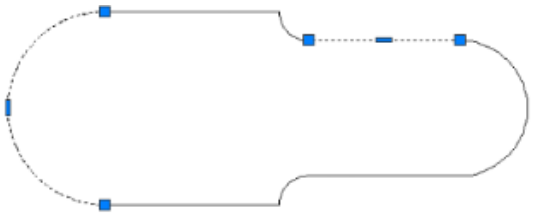
Option	Animation
Stretch or Stretch Vertex. Specify a stretch point.	
Add Vertex. Specify a point for the new vertex.	
Remove Vertex. Delete the selected vertex.	
Convert to Arc. Specify the midpoint of a straight segment to convert it to an arc segment.	
Convert to Line. Specify the midpoint of an arc segment to convert into a straight segment.	

Option	Animation
Tangent Direction. Manipulate the tangent directions to redefine the shape of a curve-fit polyline.	

Modify a Segment Within a Polyline

To select individual segments (or *subobjects*) of a polyline, press Ctrl while clicking the segments. You can use grips to modify the segments. You can use the Properties palette to change the width of individual segments; changing any other property affects the entire polyline even if only a segment is selected.

TIP The GRIPSUBOBJMODE system variable controls whether grips are automatically selected (or made "hot") when subobjects are selected.



multi-functional grips displayed on selected subobjects

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.

See also:

- Choose a Method to Modify Objects on page 776
- Modify Objects Using Grips on page 777

- Overview of Constraints on page 865
- Trim or Extend Objects on page 821
- Break and Join Objects on page 843

To modify a polyline

1. Click Home tab ➤ Modify panel ➤ Edit Polyline. 
2. Select the polyline to modify.

   NOTE To select a single arc or line segment, press Ctrl while clicking the segment.

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 an arc-fit polyline, a smooth curve consisting 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.

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.

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.

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.

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.

Quick Reference

Commands

PEDIT

Edits polylines and 3D polygon meshes.

JOIN

Joins the endpoints of linear and curved objects to create a single object.

REVERSE

Reverses the vertices of selected lines, polylines, splines, and helixes, which is useful for linetypes with included text, or wide polylines with differing beginning and ending widths.

System Variables

GRIPS

Controls the color of selected grips.

GRIPMULTIFUNCTIONAL

Specifies the access methods for multi-functional grip options.

GRIPSUBOBJMODE

Controls whether grips are automatically selected (made "hot") when subobjects are selected.

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.

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

See also:

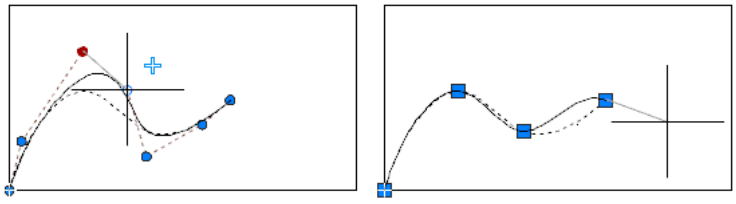
- Draw Splines on page 723
- Break and Join Objects on page 843
- Use Object Grips on page 778
- Edit NURBS Surfaces on page 1225
- Rebuild NURBS Surfaces and Curves on page 1227

Several methods are available for editing splines and changing their underlying mathematical parameters.

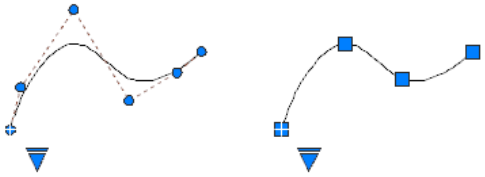
You can edit splines using multi-functional grips, SPLINEDIT, 3DEDITBAR, and the Properties palette. In addition to these operations, splines can be trimmed, extended, and filleted.

Edit Splines with Multi-Functional Grips

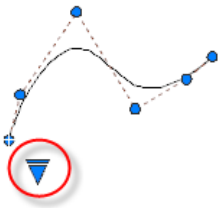
Multi-functional grips provide options that include adding control vertices and changing the tangent direction of the spline at its endpoints. Display a menu of options by hovering over a grip.



The editing options available with multi-functional grips differ depending on whether the spline is set to display *control vertices* or *fit points*. The spline on the left displays control vertices, and the one on the right displays fit points.

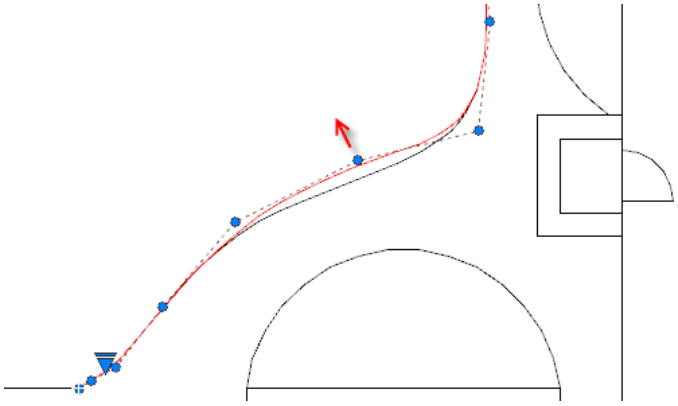


To switch between displaying control vertices and displaying fit points, click the triangular grip.

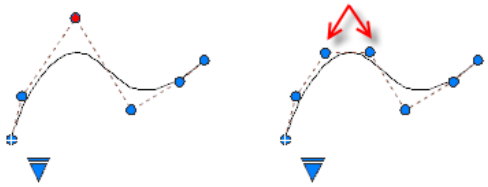


IMPORTANT Switching from displaying control vertices to fit points automatically changes the selected spline to degree 3. Splines originally created using higher-degree equations will likely change shape as a result.

In general, editing a spline with control vertices provides finer control over reshaping a small section of the curve than editing a spline with fit points.

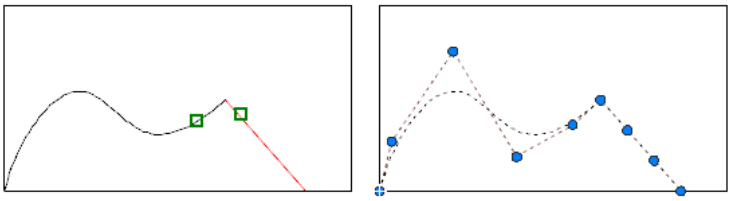


You can insert additional control vertices to a section of a spline to obtain greater control in that section at the expense of making the shape of the spline more complicated. The Refine option adds a *knot* to the spline resulting in replacing the selected control vertex with two control vertices.



Edit Splines with SPLINEDIT

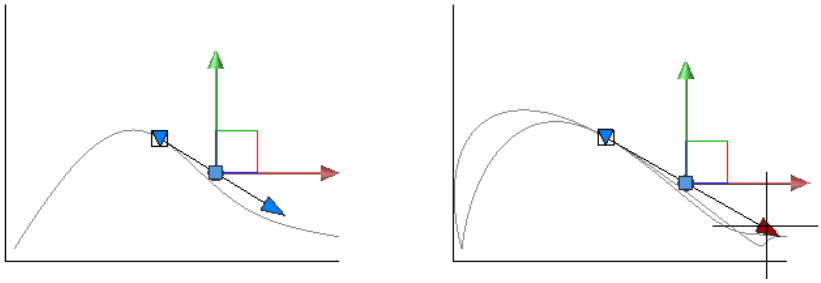
SPLINEDIT provides additional editing options, such as adding a kink to the spline, and joining a spline to another contiguous object, such as a line, arc, or other spline. As shown, objects are joined to splines with C0 continuity.



Edit Splines with 3DEDITBAR

3DEDITBAR displays a gizmo that can move a portion of a spline proportionately, or change the direction and magnitude of the tangent at a specified *base point* on the spline. To display a menu of control options, right-click the gizmo.

The gizmo in the illustration is the default setting, which is the Move Point Location option. The square grip is located at a specified base point on the spline, and is used to stretch a portion of the spline.



The red and green axis arrow grips constrain the movement of the square grip in their respective directions.

TIP Not visible in the illustration is a blue axis arrow grip that points toward you. This axis is visible in other views such as a 3D isometric view, and can be used to modify the shape of a spline in 3D.

Click the downward-pointing triangular grip to switch to the Move Tangent Direction option as illustrated below. Even though the axes of the gizmo change their location, the base point remains the same. With this option, moving the square grip changes the slope of the tangent at the base point.

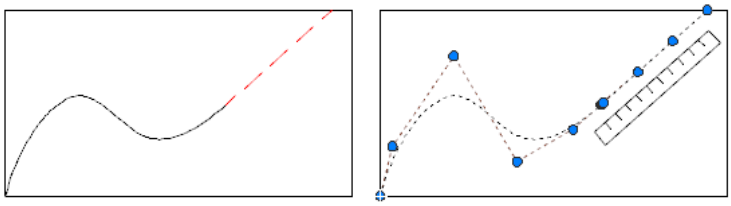
The tangent arrow grip changes the magnitude of the tangent at the base point, creating either a sharper or a flatter curvature at the base point. In the illustration, the magnitude of the tangent is being increased.

Edit Splines with a Palette

The Properties palette provides access to several spline parameters and options, including the degree of the spline, the weight for each control point, the *knot parameterization* method used in conjunction with fit points, and whether the spline is closed. For more information, see Draw Splines on page 723.

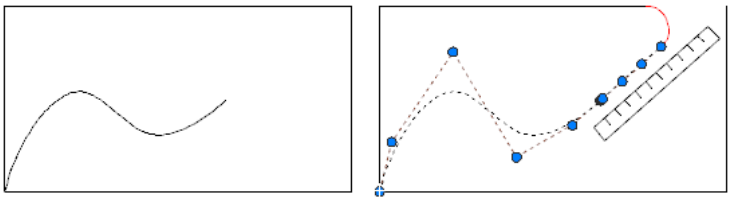
Trim, Extend, and Fillet Splines

Trimming a spline shortens it without changing the shape of the portion that remains. Extending a spline lengthens it by adding a linear portion that is tangent to the end of the spline (C1 continuity). If the shape of the spline is later changed, the tangency of the linear portion is not maintained.



Trimming a spline shortens it without changing the shape of the portion that remains.

Filleting a spline creates an arc that is tangent to the spline and the other selected object. The spline might be extended with a linear portion to complete the fillet operation.



For more information, see Modify Objects Using Grips on page 777.

To convert a spline to a polyline

1. Click Home tab ➤ Modify panel ➤ Edit Spline.
2. Select the spline to convert.
3. Enter **p** to convert to Polyline.
4. Specify a precision value or press Enter to end the command.

Quick Reference

Commands

3DEDITBAR

Reshapes splines and NURBS surfaces, including their tangency properties.

CVSHOW

Displays the control vertices for specified NURBS surfaces or curves.

CVADD

Adds control vertices to NURBS surfaces and splines.

CVHIDE

Turns off the display of control vertices for all NURBS surfaces and curves.

CVREBUILD

Rebuilds the shape of NURBS surfaces and curves.

CVREMOVE

Removes control vertices from NURBS surfaces and curves.

JOIN

Joins the endpoints of linear and curved objects to create a single object.

SPLINE

Creates a smooth curve that passes through or near a set of fit points, or that is defined by the vertices in a control frame.

SPLINEDIT

Modifies the parameters of a spline or converts a spline-fit polyline to a spline.

REVERSE

Reverses the vertices of selected lines, polylines, splines, and helixes, which is useful for linetypes with included text, or wide polylines with differing beginning and ending widths.

System Variables

GRIPS

Controls the display of grips on selected objects.

PLINECONVERTMODE

Specifies the fit method used in converting splines to 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 728

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 multiline editing commands.

Special multiline editing features are available with the MLEDIT command including the following:

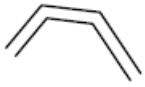
- Add or delete a vertex
- Control the visibility of corner joints
- Control the style of intersection with other multilines
- Open or close gaps in a multiline object

Add and Delete Multiline Vertices

You can add or delete any vertex in a multiline.



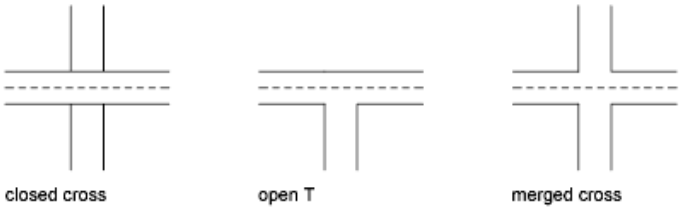
vertex in multiline to delete



multiline with vertex deleted

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 701

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.

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.

Quick Reference

Commands

MLEDIT

Edits multiline intersections, breaks, and vertices.

MLSTYLE

Creates, modifies, and manages multiline styles.

Add Constraints to Geometry

20

With parametric drawing, you can add constraints to geometry to ensure that the design conforms to specified requirements.

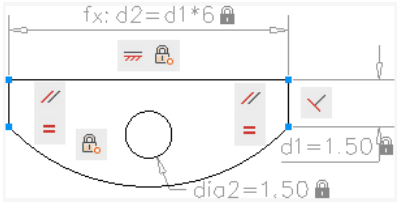
Overview of Constraints

Parametric drawing is a technology that is used for designing with constraints. *Constraints* are associations and restrictions applied to 2D geometry.

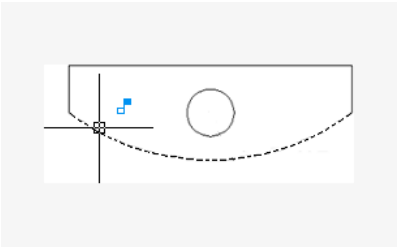
There are two general types of constraints:

- *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 976.

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 Shift 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

AUTOCONSTRAIN

Applies geometric constraints to a selection set of objects based on orientation of the objects relative to one another.

CONSTRAINTBAR

Displays or hides the geometric constraints on an object.

CONSTRAINTSETTINGS

Controls the display of geometric constraints on constraint bars.

DCDISPLAY

Displays or hides the dynamic constraints associated with a selection set of objects.

DELCONSTRAINT

Removes all geometric and dimensional constraints from a selection set of objects.

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects, or converts associative dimensions to dimensional constraints.

GEOMCONSTRAINT

Displays or hides the geometric constraints on an object.

LIST

Displays property data for selected objects.

PARAMETERS

Opens the Parameters Manager palette that includes all dimensional constraint parameters, reference parameters, and user variables in the current drawing.

PARAMETERSCLOSE

Closes the Parameters Manager palette.

TEXTEDIT

Edits a selected multiline or single-line text object, or the text in a dimension object.

CCONSTRAINTFORM

Controls whether annotational or dynamic constraints are applied to objects.

CONSTRAINTBARDISPLAY

Controls the display of constraint bars after you apply constraints and when you select geometrically constrained drawings.

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.

DYNCONSTRAINTMODE

Displays hidden dimensional constraints when constrained objects are selected.

PARAMETERCOPYMODE

Controls how constraints and referenced user parameters are handled when constrained objects are copied between drawings, Model space and layouts, and block definitions.

PARAMETERSSTATUS

Indicates whether the Parameters Manager 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

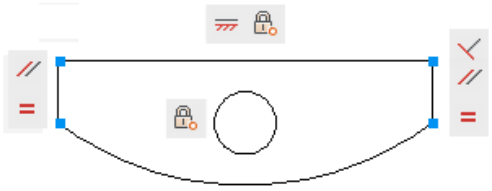
You can specify geometric constraints between 2D objects or points on objects. When you later edit the constrained geometry, the constraints are maintained.

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

NOTE The locked geometry is not associated to the other geometry without geometric constraints linked to it.



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 888

Quick Reference

Commands

AUTOCONSTRAIN

Applies geometric constraints to a selection set of objects based on orientation of the objects relative to one another.

CONSTRAINTBAR

Displays or hides the 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

Displays or hides the 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

Controls the display of constraint bars after you apply constraints and when you select geometrically constrained drawings.

CONSTRAINTNAMEFORMAT

Controls the text format for dimensional constraints.

CONSTRAINTRELAX

Indicates whether constraints are enforced or relaxed when editing an object.

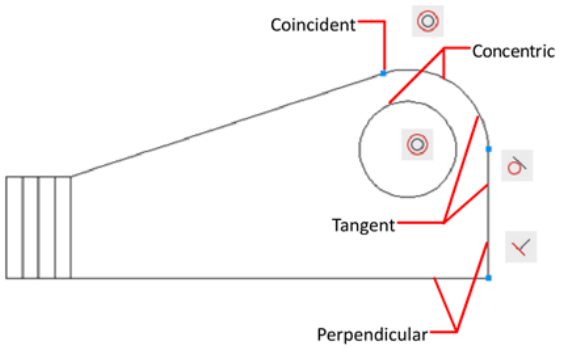
CONSTRAINTSOLVEMODE

Controls constraint behavior when applying or editing constraints.

Apply or Remove Geometric Constraints

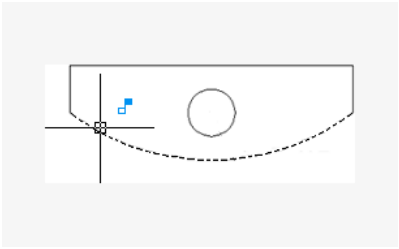
Geometric constraints associate geometric objects together, or specify a fixed location or angle.

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.

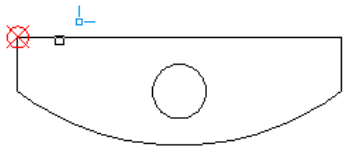
You can apply geometric constraints to 2D geometric objects only. Objects cannot be constrained between model space and paper space.

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, a 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.

The fix, horizontal, and vertical constraint icons indicate whether the constraints are applied to an object or a point.

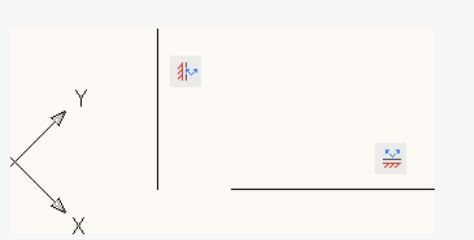
Constraint	Point	Object
Fix		
Horizontal		
Vertical		

The symmetric constraint icons indicate whether it identifying a symmetrical point or object, or the symmetrical line.

Constraint	Point	Object	Line
Symmetric			

When rolling over any icon, the constraint point markers are displayed indicating the constrained points. You do not need to roll over the icon to identify the constraints that are applied to the points of the selected object.

A different set of constraint bar icons are displayed when a horizontal or vertical constraint is not parallel or perpendicular with the current UCS.



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 Fix constraint is not applied with AUTOCONSTRAIN. You must apply the constraint individually. Equal constraint applied with AUTOCONSTRAIN resizes the selected arcs to the same radius only. It is not applied to the arc length.

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 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.

Quick Reference

Commands

CONSTRAINTBAR

Displays or hides the geometric constraints on an object.

CONSTRAINTSETTINGS

Controls the display of geometric constraints on constraint bars.

GCCOINCIDENT

Constrains two points together or a point to a curve (or an extension of a curve).

GCCOLLINEAR

Causes two or more line segments to lie along the same line.

GCCONCENTRIC

Constrains two arcs, circles, or ellipses to the same center point.

GCEQUAL

Resizes selected arcs and circles to the same radius, or selected lines to the same length.

GCFIX

Locks points and curves in position.

GCHORIZONTAL

Causes lines or pairs of points to lie parallel to the X axis of the current coordinate system.

GCPARALLEL

Causes selected lines to lie parallel to each other.

GCPERPENDICULAR

Causes selected lines to lie 90 degrees to one another.

GCSMOOTH

Constrains a spline to be contiguous and maintain G2 continuity with another spline, line, arc, or polyline.

GCSYMMETRIC

Causes selected objects to become symmetrically constrained about a selected line.

GCTANGENT

Constrains two curves to maintain a point of tangency to each other or their extensions.

GCVERTICAL

Causes lines or pairs of points to lie parallel to the Y axis of the current coordinate system.

GEOMCONSTRAINT

Displays or hides the geometric constraints on an object.

System Variables

CONSTRAINTBARMODE

Controls the display of geometrical constraints on constraint bars.

CONSTRAINTBARDISPLAY

Controls the display of constraint bars after you apply constraints and when you select geometrically constrained drawings.

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.

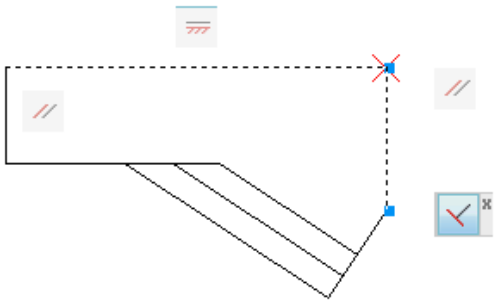


You can drag constraint bars when you need to move them out of the way, and you can also control whether they are displayed or hidden.

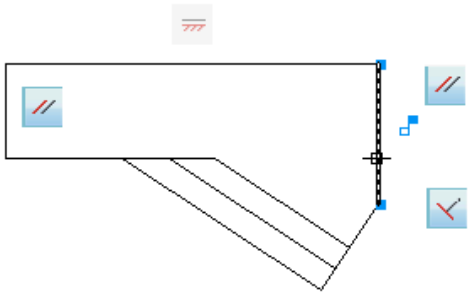
Verify the Geometric Constraints on Objects

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.

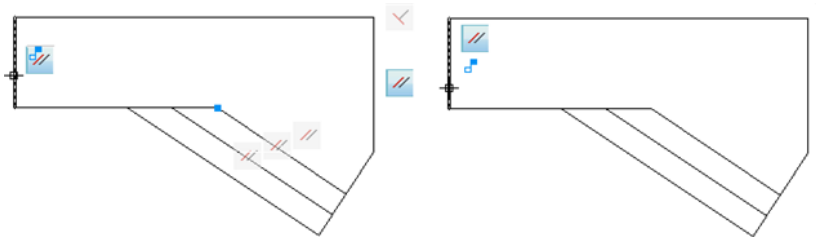
Control the Display of Constraint Bars

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
- Temporarily display the geometric constraints of the selected object

Use the Constraint Settings dialog box to control the types of geometric constraints that are displayed or hidden on constraint bars.

You can set the constraint bars to automatically and temporarily display when the constrained geometry is selected. When the geometry is no longer selected, the temporarily displayed constraint bars are hidden.



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.

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 or hide a geometric constraint

1. Click Parametric tab ➤ Geometric panel ➤ Show/Hide. 
2. Select the constrained objects.
3. Press Enter.
4. Select one of following:
   - **Show.** Displays the geometric constraints.
   - **Hide.** Hides the geometric constraints.
   - **Reset.** Displays the geometric constraints and resets the contraint bar to the default position relative to the parameters they are associated with.

To display all geometric constraints

➤ Click Parametric tab ➤ Geometric panel ➤ Show All. 

To hide all geometric constraints

➤ Click Parametric tab ➤ Geometric panel ➤ Hide All. 

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.

Quick Reference

Commands

GEOMCONSTRAINT

Displays or hides the geometric constraints on an object.

CONSTRAINTBAR

Displays or hides the 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

Controls the display of constraint bars after you apply constraints and when you select geometrically constrained drawings.

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.

Modify Objects with Geometric Constraints Applied

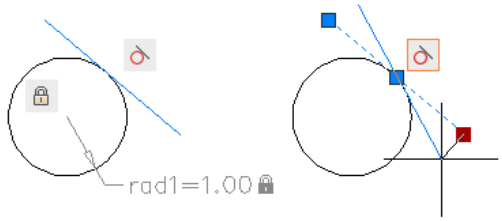
You can edit constrained geometric objects with grips, editing commands, or by relaxing or applying geometric constraints.

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

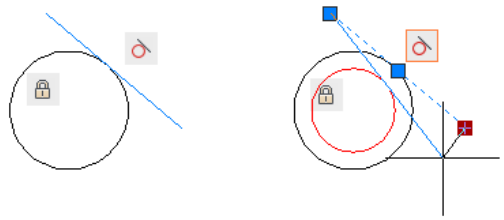
You can modify constrained geometry using grip editing modes. The geometry will maintain all applied constraints.

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, SCALE, and STRETCH 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 865.

To grip-edit constrained geometry

1. Select the constrained object.
2. Click the grips and drag it to edit the geometry.

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 Shift 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.

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.

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.

Quick Reference

Commands

GEOMCONSTRAINT

Displays or hides the geometric constraints on an object.

CONSTRAINTBAR

Displays or hides the geometric constraints on an object.

CONSTRAINTSETTINGS

Controls the display of geometric constraints on constraint bars.

System Varaibles

CONSTRAINTBARMODE

Controls the display of geometrical constraints on constraint bars.

CONSTRAINTBARDISPLAY

Controls the display of constraint bars after you apply constraints and when you select geometrically constrained drawings.

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.

Infer Geometric Constraints

You can automatically apply geometric constraints while creating and editing geometric objects.

Enabling *Infer Constraints* mode automatically applies constraints between the object you are creating or editing and the object or points associated with object snaps.

Similar to the AUTOCONSTRAIN command, constraints are applied only if the objects meet the constraint conditions. Objects are not repositioned as a result of inferring constraints.

With Infer Constraints turned on, the object snaps that you specify when creating geometry are used to infer geometric constraints. However, the following object snaps are not supported: Intersection, Apparent Intersection, Extension, and Quadrant.

The following constraints cannot be inferred:

- Fix
- Smooth

- Symmetric
- Concentric
- Equal
- Collinear

Infer Constraints with Line and Polyline

Certain object creation and editing commands can infer constraints regardless of the current object snap settings.

LINE and PLINE commands infer coincident *point-to-point* constraints. The Close option infers a coincident constraint between the start point of the first line and the endpoint of the last line.

Infer Constraints with Rectangle, Fillet, and Chamfer

The RECTANG, FILLET, and CHAMFER commands infer constraints as follows:

- RECTANG applies a pair of parallel constraints and a perpendicular constraint to the closed polyline.
- FILLET applies tangent and coincident constraints between the newly created arc and the existing trimmed or extended pair of lines.
- CHAMFER applies coincident contraints between the newly created line and the existing trimmed or extended pair of lines.

The following commands are unaffected by the Infer Constraints setting:

- SCALE
- MIRROR
- OFFSET
- BREAK
- TRIM
- EXTEND
- ARRAY

Infer Constraints with Move, Copy, and Stretch

When moving, copying, or stretching with the Infer Constraints on, you can apply coincident, perpendicular, parallel, or tangent constraints between the object being edited and the object being snapped to if the base point of the edited object is a valid constraint point of that object.

For example, if a line is stretched and snapped to an endpoint of another line, a coincident constraint is applied between the endpoints of the two lines.

A vertical or horizontal constraint can be applied between objects when you move, copy, or stretch an object from a valid constraint point while object tracking vertically or horizontally along a valid constraint point on another object.

See also:

- Use Object Snaps on page 626

To turn on and off inferred constraints

1. Click Parametric tab ➤ Geometric panel ➤ Constraint Settings, Geometric.
2. In the Constraint Settings dialog box, Geometric tab, click or clear Infer Geometric Constraints.
3. Click OK.

NOTE Alternatively, you can click the Infer Constraints button on the Status bar.

Quick Reference

Commands

CHAMFER

Bevels the edges of objects.

CONSTRAINTSETTINGS

Controls the display of geometric constraints on constraint bars.

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects, or converts associative dimensions to dimensional constraints.

FILLET

Rounds and fillets the edges of objects.

LINE

Creates straight line segments.

PLINE

Creates a 2D polyline, a single object that is composed of line and arc segments.

RECTANG

Creates a rectangular polyline.

System Variables

CONSTRAINTINFER

Controls whether the geometric constraints are inferred while drawing and editing geometry.

Constrain Distances and Angles between Objects

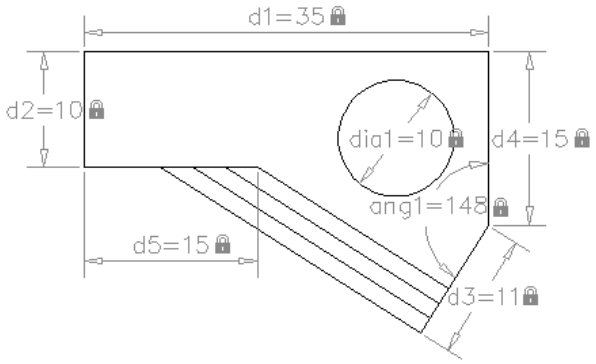
You can control distances or angles between 2D geometric objects or points on objects applying dimensional constraints and specifying values. You can also constrain geometry with variables and equations.

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

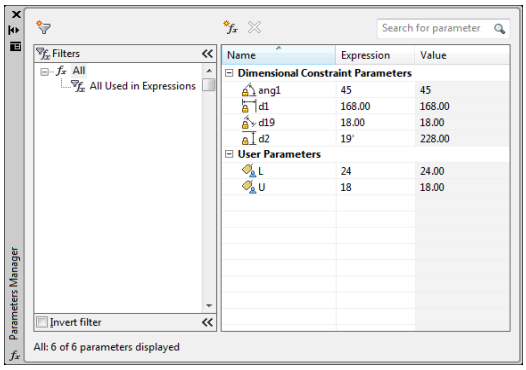
Dimensional *constraints* are different from dimension *objects* in the following ways:

- 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 outputted to a device

If you need to output a drawing with 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 892 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 903

See also:

- Overview of Geometric Constraints on page 870
- Apply Dimensional Constraints on page 892
- Constrain a Design with Formulas and Equations on page 903

Quick Reference

Commands

DCDISPLAY

Displays or hides the dynamic constraints associated with a selection set of objects.

DELCONSTRAINT

Removes all geometric and dimensional constraints from a selection set of objects.

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects, or converts associative dimensions to dimensional constraints.

LIST

Displays property data for selected objects.

PARAMETERS

Opens the Parameters Manager palette that includes all dimensional constraint parameters, reference parameters, and user variables in the current drawing.

-PARAMETERS

Opens the Parameters Manager palette that includes all dimensional constraint parameters, reference parameters, and user variables in the current drawing.

PARAMETERSCLOSE

Closes the Parameters Manager palette.

TEXTEDIT

Edits a selected multiline or single-line text object, or the text in a dimension 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.

DYNCONSTRAINTMODE

Displays hidden dimensional constraints when constrained objects are selected.

PARAMETERCOPYMODE

Controls how constraints and referenced user parameters are handled when constrained objects are copied between drawings, Model space and layouts, and block definitions.

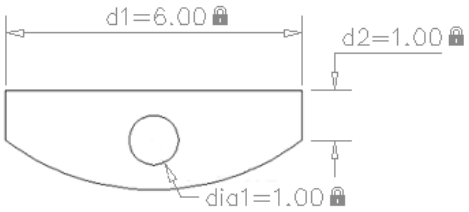
PARAMETERSSTATUS

Indicates whether the Parameters Manager 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 parameter*.

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

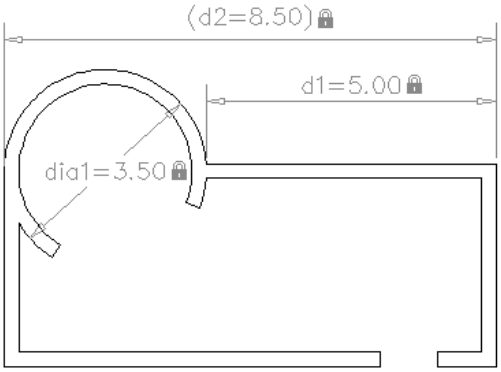
NOTE To display the text used in annotational constraints in the same format as used in dimensions, set the CONSTRAINTNAMEFORMAT system variable to 1.

After plotting, you can use the Properties palette to convert annotational constraints back to dynamic constraints.

Reference Parameters

A reference parameter 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 parameters 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 parameter, *d2*, displays the total width but does not constrain it. The textual information in reference parameters 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 parameter.

NOTE You cannot change a reference parameter back to a dimensional constraint if doing so would overconstrain the geometry.

To convert associative dimensions to dimensional constraints

1. Click Parametric tab ➤ Dimensional panel ➤ Convert.
2. Select the associative dimensions you want to convert.
3. Press Enter.

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.

Quick Reference

Commands

DCALIGNED

Constrains the distance between two points on different objects.

DCANGULAR

Constrains the angle between line or polyline segments, the angle swept out by an arc or a polyline arc segment, or the angle between three points on objects.

DCCONVERT

Converts associative dimensions to dimensional constraints.

DCDIAMETER

Constrains the diameter of a circle or an arc.

DCDISPLAY

Displays or hides the dynamic constraints associated with a selection set of objects.

DCFORM

Specifies whether the dimensional constraint being created is dynamic or annotational.

DCHORIZONTAL

Constrains the X distance between points on an object, or between two points on different objects.

DCLINEAR

Creates a horizontal, vertical, or rotated constraint based on the locations of the extension line origins and the dimension line.

DCRADIUS

Constrains the radius of a circle or an arc.

DCVERTICAL

Constrains the Y distance between points on an object, or between two points on different objects.

DELCONSTRAINT

Removes all geometric and dimensional constraints from a selection set of objects.

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects, or converts associative dimensions to dimensional constraints.

LIST

Displays property data for selected objects.

PARAMETERS

Opens the Parameters Manager palette that includes all dimensional constraint parameters, reference parameters, and user variables in the current drawing.

-PARAMETERS

Opens the Parameters Manager palette that includes all dimensional constraint parameters, reference parameters, and user variables in the current drawing.

PARAMETERSCLOSE

Closes the Parameters Manager palette.

TEXTEDIT

Edits a selected multiline or single-line text object, or the text in a dimension 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.

DYNCONSTRAINTMODE

Displays hidden dimensional constraints when constrained objects are selected.

PARAMETERCOPYMODE

Controls how constraints and referenced user parameters are handled when constrained objects are copied between drawings, Model space and layouts, and block definitions.

PARAMETERSSTATUS

Indicates whether the Parameters Manager 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 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 DCDISPLAY command.

By default, if you select an object associated with a hidden dynamic constraint, all dynamic constraints associated with that object are temporarily displayed.

You can display or hide the dynamic constraints for all objects or for a selection set.

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 or hide dynamic dimensional constraints

1 Click Parametric tab ➤ Dimensional panel ➤ Show/Hide. 

2 Select the constrained objects.

3 Press Enter.

4 Select one of following:

- **Show.** Displays the dimensional constraints.
- **Hide.** Hides the dimensional constraints.

Quick Reference

Commands

DCDISPLAY

Displays or hides the dynamic constraints associated with a selection set of objects.

DELCONSTRAINT

Removes all geometric and dimensional constraints from a selection set of objects.

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects, or converts associative dimensions to dimensional constraints.

LIST

Displays property data for selected objects.

PARAMETERS

Opens the Parameters Manager palette that includes all dimensional constraint parameters, reference parameters, and user variables in the current drawing.

-PARAMETERS

Opens the Parameters Manager palette that includes all dimensional constraint parameters, reference parameters, and user variables in the current drawing.

PARAMETERSCLOSE

Closes the Parameters Manager palette.

TEXTEDIT

Edits a selected multiline or single-line text object, or the text in a dimension 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.

DYNCONSTRAINTMODE

Displays hidden dimensional constraints when constrained objects are selected.

PARAMETERCOPYMODE

Controls how constraints and referenced user parameters are handled when constrained objects are copied between drawings, Model space and layouts, and block definitions.

PARAMETERSSTATUS

Indicates whether the Parameters Manager 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 edit the names, values, and expressions that are associated with dimensional constraints using in-place editing:

- 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

You can reference other dimensional constraints by selecting them during an in-place editing operation.

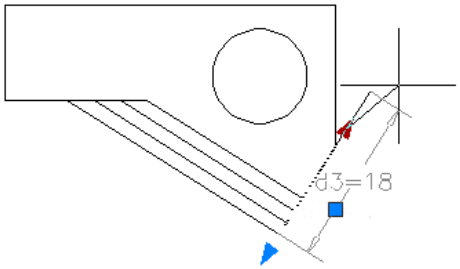
NOTE You cannot edit the Expression and Value properties for a reference parameter.

Modify Dimensional Constraints Using Their Grips

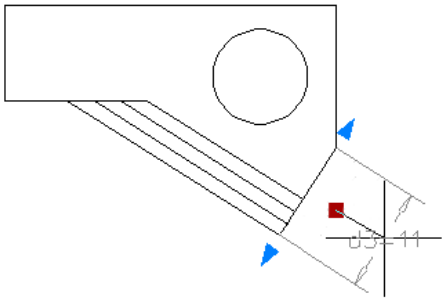
You can modify a constrained object either by using the triangular grips or the square grips on the associated dimensional constraint.

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 865.

See also:

- Overview of Constraints on page 865

To grip-edit a dimensional constraint

1. Select a constrained object.
2. Click the grips and drag to edit the geometry.

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.

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.

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 Shift key.
5. Move the object to the desired location.

The constraint is relaxed for the object, and you should be able to move it.

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.

Quick Reference

Commands

DCDISPLAY

Displays or hides the dynamic constraints associated with a selection set of objects.

DELCONSTRAINT

Removes all geometric and dimensional constraints from a selection set of objects.

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects, or converts associative dimensions to dimensional constraints.

LIST

Displays property data for selected objects.

PARAMETERS

Opens the Parameters Manager palette that includes all dimensional constraint parameters, reference parameters, and user variables in the current drawing.

-PARAMETERS

Opens the Parameters Manager palette that includes all dimensional constraint parameters, reference parameters, and user variables in the current drawing.

PARAMETERSCLOSE

Closes the Parameters Manager palette.

TEXTEDIT

Edits a selected multiline or single-line text object, or the text in a dimension 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.

DYNCONSTRAINTMODE

Displays hidden dimensional constraints when constrained objects are selected.

PARAMETERCOPYMODE

Controls how constraints and referenced user parameters are handled when constrained objects are copied between drawings, Model space and layouts, and block definitions.

PARAMETERSSTATUS

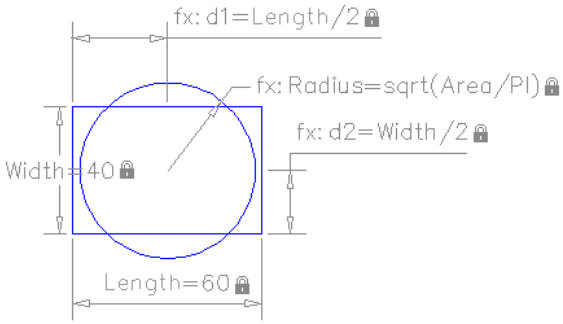
Indicates whether the Parameters Manager is displayed or hidden.

Constrain a Design with Formulas and Equations

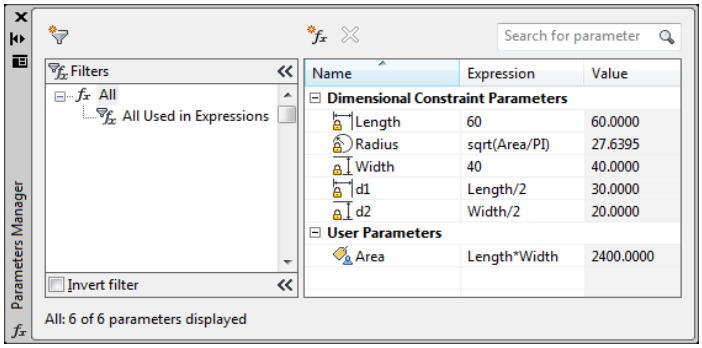
You can control geometry using mathematical expressions that include the names of dimensional constraints, user variables, and functions.

Overview of Formulas and Equations

Formulas and equations can be represented either as expressions within dimensional constraint parameters or by defining 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 constraint parameters are set to constants. The *d1* and *d2* constraints are simple expressions that reference the *Length* and *Width*. The *Radius* dimensional constraint parameter 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 parameter 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 parameter, defined in a user variable, or some other combination.

Protect Expressions in Dynamic Constraints

When a *dynamic dimensional constraint* references one or more parameters, the prefix *fx:* is added to the name of the constraint. This prefix is displayed only in the drawing. Its purpose is to help you avoid accidentally overwriting parameters and formulas when the *dimension name format* is set to *Value* or *Name*, which suppresses the display of the parameters and formulas.

Quick Reference

Commands

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects, or converts associative dimensions to dimensional constraints.

PARAMETERS

Opens the Parameters Manager palette that includes all dimensional constraint parameters, reference parameters, and user variables in the current drawing.

-PARAMETERS

Opens the Parameters Manager palette that includes all dimensional constraint parameters, reference parameters, and user variables in the current drawing.

Control Geometry with the Parameters Manager

The Parameters Manager lists dimensional constraint parameters, reference parameters, and user variables, which you can also create, edit, and organize.

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 parameter 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 parameter or user variable.

- Click a column heading to sort the list of parameters by name, expression, or value.

Use Operators in Expressions

Dimensional constraint parameters 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

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**".

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: (1) unary negation, (2) exponents, (3) multiplication and division, and (4) addition and subtraction
3. Operators of equal precedence from left to right

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*)
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*)

Function	Syntax
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 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.

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.

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.

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.

Quick Reference

Commands

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects, or converts associative dimensions to dimensional constraints.

PARAMETERS

Opens the Parameters Manager palette that includes all dimensional constraint parameters, reference parameters, and user variables in the current drawing.

-PARAMETERS

Opens the Parameters Manager palette that includes all dimensional constraint parameters, reference parameters, and user variables in the current drawing.

Organize Parameters into Groups

Organize dimensional and user-defined parameters into groups, and control whether they are displayed in the Parameters Manager list.

When many dimensional and user-defined parameters are defined in a drawing, it can be useful to use the Parameters Manager to create several *parameter groups*, and then assign the parameters to one or more of the groups with a simple drag and drop operation. The result lets you view *one* group of parameters at a time, organizing and limiting their display in the Parameters Manager. Thus, parameter groups are used as a display *filter* for the parameter list.

Expanding the *filter tree* in the Parameters Manager displays all group filters created in the current space, Model space or a layout. There are two predefined filters displayed in the filter tree that cannot be edited:

- **All.** Lists all parameters in the current space.
- **All Used in Expressions.** Lists all parameters used in expressions or defined by an expression.

When you use the Invert Filter option, it will display all the parameters **not** in the group instead of displaying only the parameters belonging to the group.

Search For Parameters

You can enter characters and wildcards such as * in the edit box of the Parameters Manager to search for parameters by name. This immediately filters the parameter list as you enter the characters.

To create, modify, or delete a parameter group

1. Click Parametric tab ➤ Manage panel ➤ Parameters Manager. 
2. In the Parameters Manager, you can do the following:
   - **Create a parameter group.** Click the Parameter Group button and specify a group name.
   - **Rename a parameter group.** Right-click the parameter group, choose Rename, and specify a new group name.
   - **Delete a parameter group.** Right-click the parameter group and choose Delete.

To add or remove parameters to or from parameter groups

1. Click Parametric tab ➤ Manage panel ➤ Parameters Manager. 
2. In the Parameters Manager, you can do the following:
   - **Add parameters to any user-defined parameter group.** Select the parameters that you want to add to a parameter group. Drag and drop the selected parameters into a parameter group in the filter tree. A parameter can belong to multiple groups.
   - **Remove parameters from a group.** Select the parameters you want to remove. Right-click the selected parameters and choose Remove from Group Filter.

Quick Reference

Commands

PARAMETERS

Opens the Parameters Manager palette that includes all dimensional constraint parameters, reference parameters, and user variables in the current drawing.

-PARAMETERS

Opens the Parameters Manager palette that includes all dimensional constraint parameters, reference parameters, and user variables in the current drawing.

PARAMETERSCLOSE

Closes the Parameters Manager palette.

Part 7: Define and Reference Blocks

Work with Blocks 21

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.

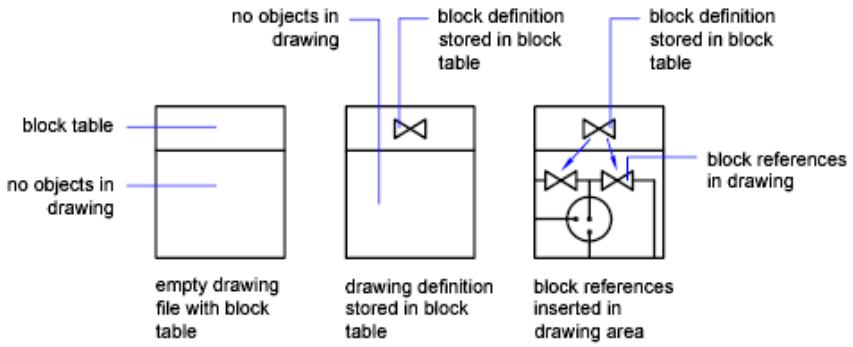
Overview of Blocks

How Blocks Are Stored and Referenced

Every drawing file has a block definition table that stores all block definitions, which consist of all information associated with the block. It is these block definitions that are referenced when you insert blocks in your drawing.

Each rectangle below represents a separate drawing file and is divided into two parts:

- The block definition table
- The objects in the drawing



When you insert a block you are inserting a block reference. The information is not 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.

Use PURGE to remove unused block definitions from a drawing.

Blocks and Layers

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.

Annotative Blocks

You can also create annotative on page 2289 blocks. For more information about creating and working with an annotative blocks, see Create Annotative Blocks and Attributes on page 1346.

See also:

- Scale Annotations on page 1324
- Create Annotative Blocks and Attributes on page 1346

Quick Reference

BLOCK

Creates a block definition from selected objects.

PURGE

Removes unused items, such as block definitions and layers, from the drawing.

WBLOCK

Saves selected objects or converts a block to a specified drawing file.

MAXSORT

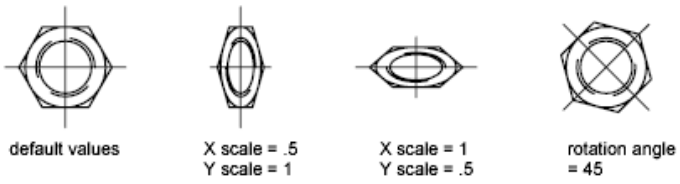
Sets the maximum number of symbol names or block names sorted by listing commands.

Insert Blocks

When you insert a block, you create a block reference and specify its location, scale, and rotation.

Scale Block References

You can specify the scale of a block reference using different *X, Y,* and *Z* values.



A block that uses different drawing units than the units specified for the drawing is automatically scaled by a factor equivalent to the ratio between the two units.

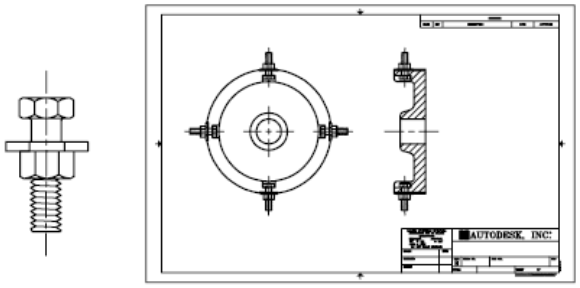
Edit Custom Properties and Attribute Values

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.

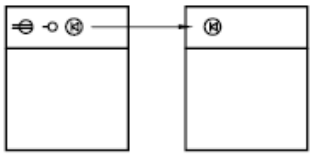
When a block is dragged from a tool palette into a drawing, it is scaled automatically according to the ratio of units defined in the block and defined in the current drawing. For example, if the current drawing uses meters as its units and a block is defined using centimeters as its units, the ratio 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.



block definition inserted from block library drawing

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.

See also:

- Create Drawing Files for Use as Blocks on page 931
- Overview of Blocks on page 915
- Create Block Libraries on page 937
- Work with Dynamic Blocks in Drawings on page 922
- Add Text and Blocks to Tables on page 1514
- Add Content with DesignCenter on page 91
- Create and Use Tools from Objects and Images on page 62

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.

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.

To change properties of a block as you insert it

1 Click View tab ➤ Palettes panel ➤ Properties.

2 Click Home tab ➤ Block panel ➤ Insert.

3 In the Insert dialog box, in the Name box, select a name from a list of block definitions.

4. Select Specify On-Screen to use the pointing device to specify one or more of the following:
   - Insertion Point
   - Scale
   - Rotation
5. Click OK.
6. In the Properties palette, change the properties of the block (or dynamic block).

Quick Reference

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.

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

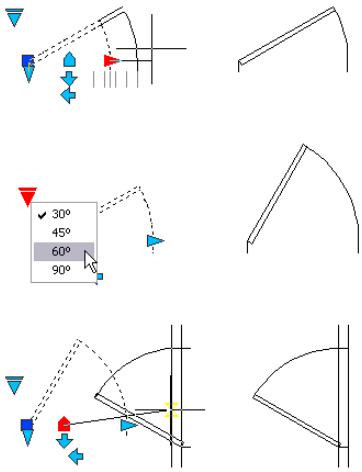
Sets target drawing units value when INSUNITS is set to 0.

Work with Dynamic Blocks in Drawings

A dynamic block reference can be changed in a drawing while you work.

Overview of Dynamic Blocks

Dynamic block references contain grips or custom properties that change the way the reference is displayed in the drawing after it is inserted. For example, a dynamic block reference of a door can change size after you insert the block reference into your drawing. Dynamic blocks allow you to insert one block that can change shape, size, or configuration, instead of inserting one of many static block definitions.



Work with Action Parameters

Dynamic blocks that contain action parameters display grips that are associated with a point, object, or region in the block definition. When you edit the grip, an associated action is triggered that changes the way the block reference is displayed.

You can hover over a grip to display a tooltip or prompt that explains the parameter related to the grip. The display of the tooltip is controlled by the GRIPTIPS system variable.

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.

Quick Reference

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.

BTMARKDISPLAY

Controls whether or not value set markers are displayed for dynamic block references.

GRIPTIPS

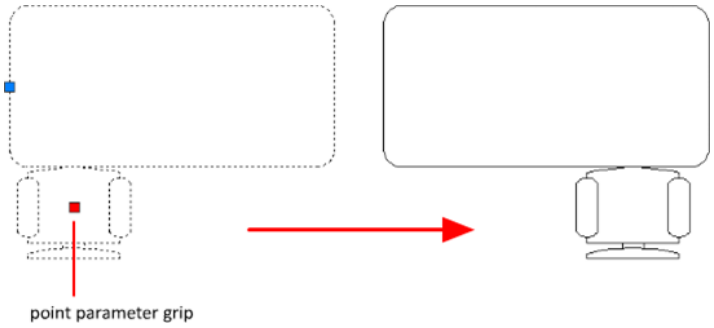
Controls the display of grip tips when the cursor hovers over grips on dynamic blocks and custom objects that support grip tips.

Work With Action Parameters in Blocks

Use grips or the Properties palette to manipulate a block reference that contains action parameters.

Use Grips to Change Blocks Containing Action Parameters

You can manipulate a block that contains action parameters with custom grips. For example, when you drag the grip on the chair in the block reference below, the chair moves.



The following table shows the different types of custom grips that can be included in a dynamic block.

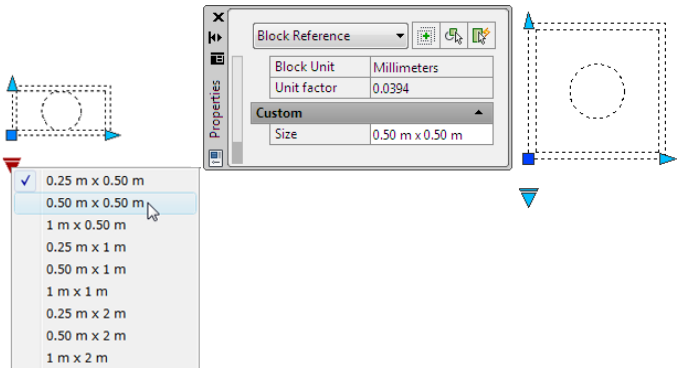
Grip Type	How the Grip Can Be Manipulated in a Drawing
Standard	Within a plane in any direction
Linear	Back and forth in a defined direction or along an axis
Rotation	Around an axis
Flip	Clicked to flip the dynamic block reference
Alignment	Within a plane in any direction; when moved over an object, triggers the block reference to align with the object
Lookup	Clicked to display a list of items

Work with Custom Properties

When you select a dynamic block reference, custom properties are listed in the Properties palette under Custom. When you change the value of the custom property, the block reference is updated accordingly.

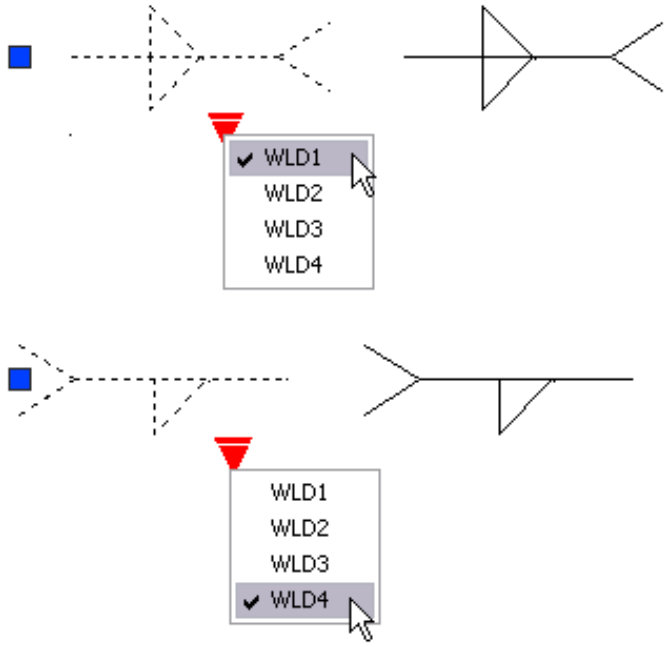
Work With Lookup Grips

A block reference that contains a lookup grip allows you to specify a preset value that changes the way the block reference is displayed. The new size is displayed in the Properties palette under Custom.



Control Visibility of Block References

A block definition can contain a visibility state grip, which determines several graphical representations of the same block reference.



Reset a Block to Display Default Geometry

When you reset a block reference, the block changes back to the default specified in the block definition. For example, you can make a block dynamic again if you non-uniformly scale or explode a dynamic block reference.

Quick Reference

RESETBLOCK

Resets one or more dynamic block references to the default values of the block definition.

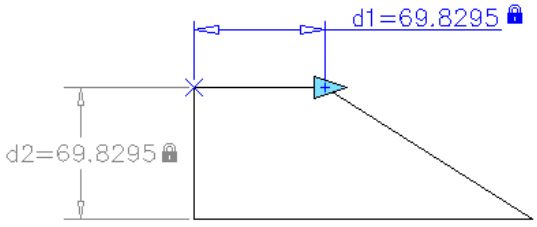
GRIPDYNCOLOR

Controls the color of custom grips for dynamic blocks.

Work With Constraint Parameters in Blocks

Use the Parameters Manager to manipulate a block reference that contains constraint parameters.

Constraint parameters are authored with mathematical expressions that affect the geometry of the block reference. They display dynamic, editable custom properties that can be manipulated outside of the Block Editor, similar to action parameters.



block reference with constraint (gray) and constraint parameter (blue, with grip)

When you select a block reference that contains constraint parameters, the editable parameters are listed in the Parameters Manager. When you change the value of the parameter, the block reference is updated accordingly.

Quick Reference

PARAMETERS

Opens the Parameters Manager palette that includes all dimensional constraint parameters, reference parameters, and user variables in the current drawing.

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 915

To remove a block definition

1. Click File ➤ Drawing Utilities ➤ 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.

Quick Reference

PURGE

Removes unused items, such as block definitions and layers, from the drawing.

Create and Modify Blocks 22

A block definition is a set of objects that are grouped together as one named object with a base point and unique properties.

Define Blocks

You create blocks by associating objects and giving them a name.

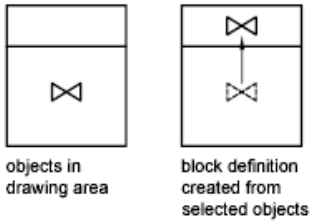
Create Blocks Within a Drawing

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.

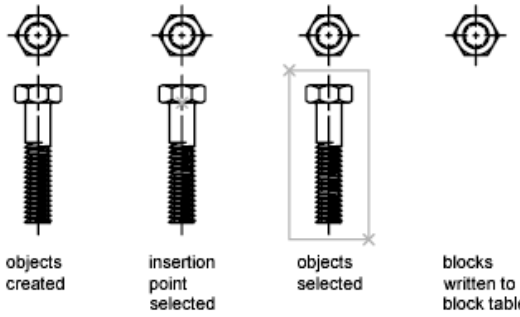
Each block definition includes a block name, one or more objects, the coordinate values of the base point to be used for inserting the block, and any associated attribute data.

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 915.



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.

See also:

- Overview of Blocks on page 915
- Overview of the Block Editor on page 968

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.

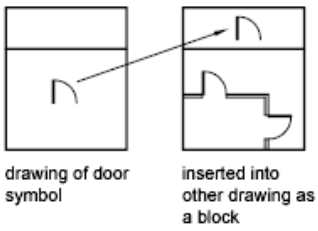
Quick Reference

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 on page 1771.

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.

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.

Quick Reference

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

Saves selected objects or converts a block to a specified drawing file.

Control the Color and Linetype Properties in Blocks

Assign Color and Linetype Properties

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.

Change the Color and Linetype in a Block

You can change the color and linetype of the objects within a block only if the objects in that block were created with floating properties.

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 Properties of Objects on page 507

Quick Reference

COLOR

Sets the color for new objects.

LAYER

Manages layers and layer properties.

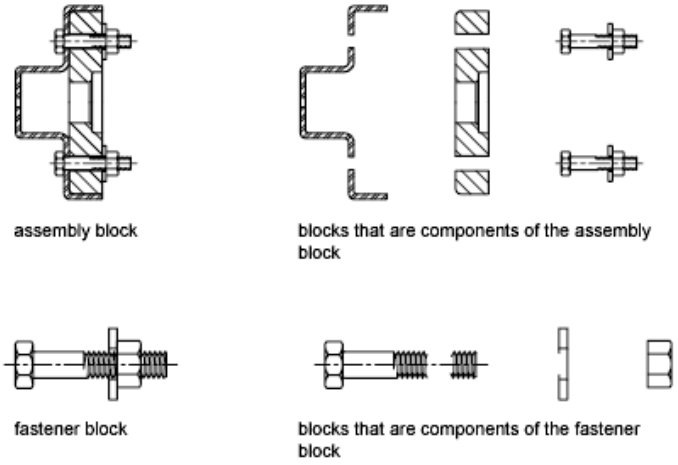
LINETYPE

Loads, sets, and modifies linetypes.

PROPERTIES

Controls properties of existing objects.

Nest Blocks



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.

See also:

- Overview of Blocks on page 915
- Add Constraints to Dynamic Blocks on page 976

Quick Reference

BLOCK

Creates a block definition from selected objects.

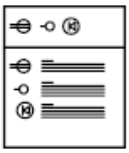
Create Block Libraries

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.

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).

Quick Reference

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 insert the block reference 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 62.

Quick Reference

TOOLPALETTES

Opens the Tool Palettes window.

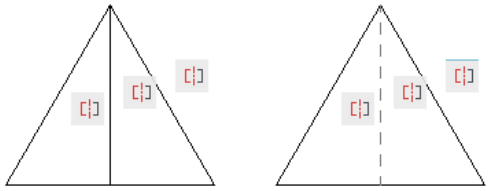
Create Construction Geometry Within a Block

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 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 730
- Add Constraints to Geometry on page 865

Quick Reference

BCONSTRUCTION

Converts geometry into construction geometry.

BVSTATE

Creates, sets, or deletes a visibility state in a dynamic block.

BLOCKTESTWINDOW

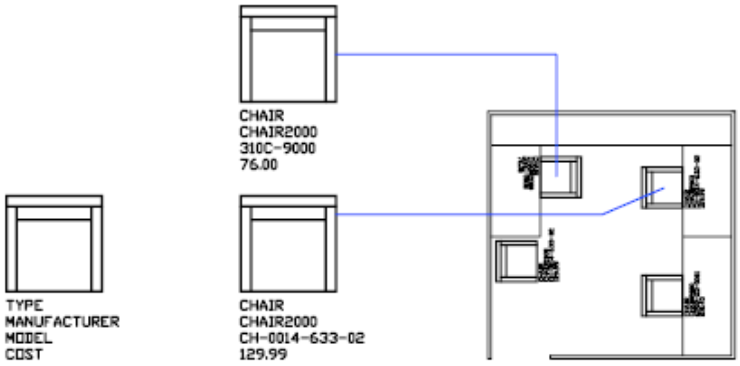
Indicates whether or not a test block window is current.

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 annotative on page 2289 attributes. For more information about creating and working with an annotative attributes, see Create Annotative Blocks and Attributes on page 1346.

See also:

- Modify a Block Attribute Definition on page 957

- Modify the Data in Block Attributes on page 956
- Scale Annotations on page 1324

Quick Reference

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.

AFLAGS

Sets options for attributes.

ATTDIA

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

ATTIPE

Controls the which Text Formatting toolbar is used with the in-place editor for modifying 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, 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 message box that lets you cancel the operation.

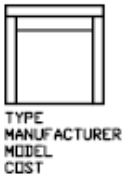
Correct Mistakes in Block Attribute Definitions

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 (BATTORDER command) 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. (Optional) Under Text Settings, Boundary Width, specify a value.
4. (Optional) Click the Multiline In-Place Text Editor button to use the in-place text editor to format the attribute in the drawing.
5. 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.

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.

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.

Quick Reference

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.

AFLAGS

Sets options for attributes.

ATTIPE

Controls the which Text Formatting toolbar is used with the in-place editor for modifying 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 on page 1969

Quick Reference

DATAEXTRACTION

Extracts drawing data and merges data from an external source to a data extraction table or external file.

DXEVAL

Controls when data extraction tables are compared against the data source, and if the data is not current, displays an update notification.

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.

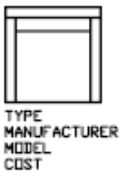
The following template file includes the 15 possible fields. *N* means numeric, *C* means character, *www* means a 3 digit number for the total width of the field, and *ddd* means a 3 digit number representing how many numeric decimal places are to be displayed to the right of the decimal point.

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 delimiter is a comma. The following two template records can be used to override these defaults:

C:QUOTE c (Character string delimiter)
C:DELIM c (Field delimiter)

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
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. See the documentation for your spreadsheet program for information about how to use data from other applications. If you open the file in a text editor or a word 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 947 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.

Quick Reference

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 939
- Modify Dynamic Block Definitions on page 974

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.

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.

Quick Reference

BLOCK

Creates a block definition from selected objects.

EXPLODE

Breaks a compound object into its component objects.

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

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 953

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

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.

ATTIPE

Controls the which Text Formatting toolbar is used with the in-place editor for modifying 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 942
- Modify a Block Definition on page 953

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.

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.

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.

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).

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.

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.

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.

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.

Quick Reference

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.

ATTIPE

Controls the which Text Formatting toolbar is used with the in-place editor for modifying 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

When you explode a block reference, the block reference is disassembled into its component objects; however, the block definition still exists in the drawing for insertion later.

You can automatically explode block references as you insert them by selecting the Explode option in the Insert dialog box.

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

EXPLODE

Breaks a compound object into its component objects.

XPLODE

Breaks a compound object into its component objects.

Add Behaviors to Blocks (Dynamic Blocks)

23

Instead of being a fixed part of a drawing, a dynamic block reference can be changed or manipulated as you work in a drawing.

Overview of Dynamic Blocks

Dynamic block definitions allow you to edit a block reference outside of the Block Editor. They contain rules, or parameters, for how to change the appearance of the block reference when it is inserted in the drawing.

A dynamic block reference inserts one block that can be edited in several different ways. For example, instead of creating multiple interior door blocks of different sizes, you can create one resizable door block.

You author dynamic blocks with either constraint parameters or action parameters.

NOTE Using both constraint parameters and action parameters in the same block definition is not recommended.

Constrain Block Definitions

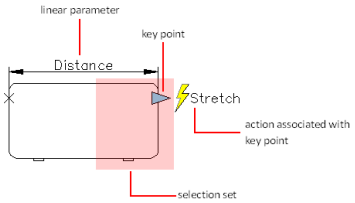
In a block definition, constraint parameters

- Associate objects with one another
- Restrict geometry or demensions

Once the block is inserted into the drawing, the constraint parameters can be edited as properties by using the Properties Manager.

Add Actions and Parameters to Block Definitions

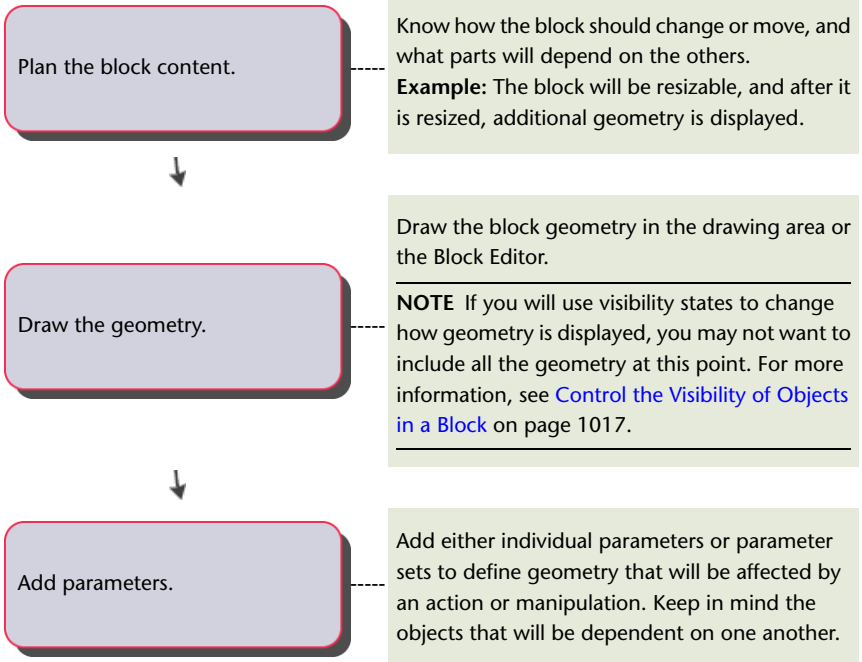
In a block definition, actions and parameters provide rules for the behavior or a block once it is inserted into the drawing.

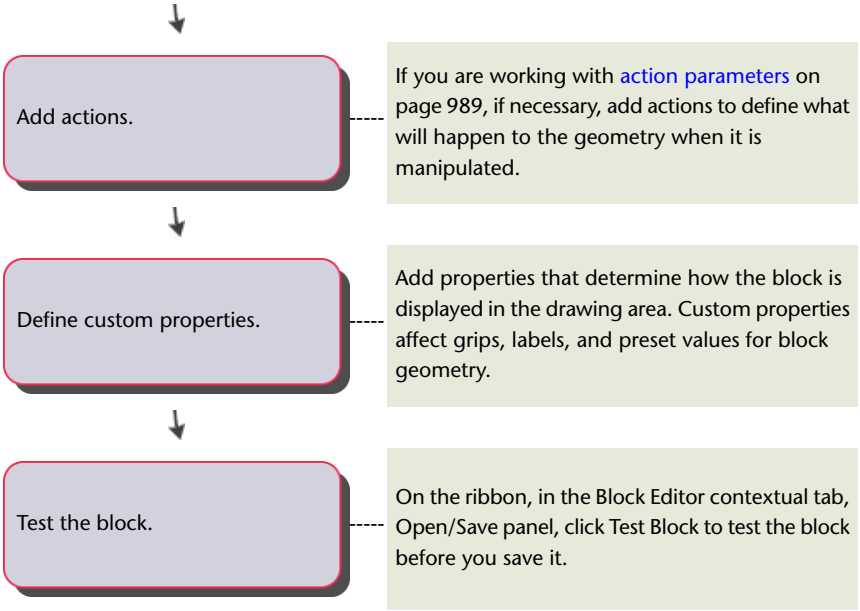


Depending on the specified block geometry, or parameter, you can associate an action to that parameter. The parameter is represented as a grip in the drawing. When you edit the grip, the associated action determines what will change in the block reference.

Just as with constraint parameters, action parameters can have editable properties that are changed using the Properties Manager.

Quick Start to Creating Dynamic Blocks





Quick Reference

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.

Create and Edit Dynamic Blocks

The Block Editor provides tools for adding dynamic behavior to block definitions.

Overview of the Block Editor

In the Block Editor, you can

- Define a block
- Add a geometric or dimensional constraint
- Add an action parameter
- Define attributes
- Manage visibility states
- Test and save the block definition

UCS in the Block Editor

Within 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. Keep the following in mind:

- While you can assign parameters to an existing 3D block definition, the block reference cannot be edited along the *Z* axis.
- While you can add actions to a dynamic block that contains solid objects, 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).

NOTE The Command window should be displayed in the Block Editor, as it displays prompts for many aspects of creating dynamic blocks.

See also:

- Define Block Attributes on page 942

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.

To create a new block definition in the Block Editor

1 Click Home tab ➤ Block panel ➤ Create. 

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.

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.

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.

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.

Quick Reference

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.

BLOCKEDITLOCK

Disallows opening of the Block Editor and editing of dynamic block definitions.

Create Custom Block Authoring Tools

Use the Block Authoring palettes to create custom tools for authoring dynamic blocks.

Similar to tool palettes, you can also use these palettes to add parameters and actions to your dynamic block definition.

Change Properties of Tools

In the Tool Properties dialog box, you can change the following properties of custom tools:

- Tool description
- Parameter type
- Associated action(s)
- Key point on the parameter to which the action is tied (if applicable)
- Tool palette image

NOTE You cannot drag parameters and actions from the Block Editor onto a regular tool palette.

Use Parameter Sets

The Parameter Sets tab of the Block Authoring palettes allows you to add commonly paired parameters and actions to your dynamic block definition.

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.

See also:

- Add Parameters to Dynamic Blocks on page 988
- Overview of Actions on page 989
- Control Tool Properties on page 72
- Organize Tool Palettes on page 79

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 or remove an action from 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 either Add or Delete.
8. (Optional) Repeat steps 3 and 4 to add or remove additional actions.
9. Click OK.
10. In the Tool Properties dialog box, click OK.

Quick Reference

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.

BLOCKEDITOR

Indicates whether or not the Block Editor is open.

Test Blocks Within the Block Editor

Test your block definition before you save or exit the Block Editor.

The test block window reflects the current block definition in the Block Editor. The test block window closes when you

- Open a different block definition
- Save the current block definition with a different name
- Exit the Block Editor

NOTE If you exit AutoCAD while in the Test Block window, you will not be prompted to save the file.

Quick Reference

BTESTBLOCK

Displays a window within the Block Editor to test a dynamic block.

BLOCKTESTWINDOW

Indicates whether or not a test block window is current.

Modify Dynamic Block Definitions

Correct Errors in Action Parameters

A yellow alert icon is displayed when

- A parameter is not associated with an action
- An action is not associated with a parameter or selection set



To correct these errors, hover over the yellow alert icon until the tooltip displays a description of the problem. Then double-click the constraint and follow the prompts.

Save Dynamic Blocks

When you save a block definition, the current values of the geometry and parameters in the block become the default values for the block reference. 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.

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 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. Click OK.
4. To save the block definition in the drawing, click File menu ➤ Save.

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. Click OK.
5. In the Browse For Drawing File dialog box, click Save.

Quick Reference

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.

BCLOSE

Closes the Block Editor.

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.

BSAVE

Saves the current block definition.

BSAVEAS

Saves a copy of the current block definition under a new name.

BVSTATE

Creates, sets, or deletes a visibility state in a dynamic block.

Add Constraints to Dynamic Blocks

You can add geometric and dimensional constraints to a dynamic block.

Overview of Constraints in Dynamic Blocks

When you add constraints to a dynamic block definition, you have the option of adding editable properties that control the parameters once they are inserted in a drawing.

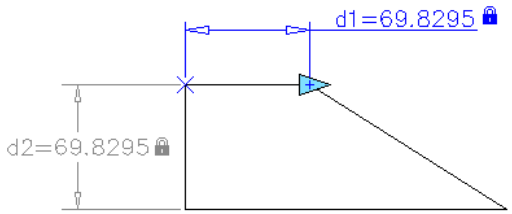
Geometric Constraints in Block Definitions

You can add geometric constraints to blocks the same way you would to geometry outside of the Block Editor. Geometric constraints are available both on the ribbon and on the Block Authoring palettes.

Dimensional Constraints in Block Definitions

A dimensional constraint applied within the Block Editor using the BCPARAMETER command is called a *constraint parameter*. 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 that can be displayed or edited for the block reference.



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:

- Constrain Objects Geometrically on page 870
- Apply Dimensional Constraints on page 892
- Specify Properties for Dynamic Blocks on page 1003
- Associative Dimensions on page 1527

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.

Quick Reference

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, or converts associative dimensions to dimensional constraints.

BCONSTATUSMODE

Turns the constraint display status on and off and controls the shading of objects based on their constraint level.

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 - Block Editor.

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 - Block Editor
- Constrain a Design with Formulas and Equations on page 903
- Specify Properties for Dynamic Blocks on page 1003

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.

Quick Reference

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, or converts associative dimensions to dimensional constraints.

BCONSTATUSMODE

Turns the constraint display status on and off and controls the shading of objects based on their constraint level.

Control Constrained Blocks with the Parameters Manager

The Parameters Manager allows you to display and edit constraints, user parameters, action parameters, and block attributes from within the Block Editor.

Within the Block Editor, the Parameters Manager displays and controls the following categories:

- Action Parameters
- Dimensional Constraint Parameters
- Reference Parameters
- User Parameters
- Attributes

For each of the above categories, you can display and control the following:

- Name
- Expression
- Value
- Type

- Show or Hide Information
- Display Order
- Description

For more information on the Parameters Manager in the Block Editor, see Parameters Manager - Block Editor.

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

PARAMETERS

Opens the Parameters Manager palette that includes all dimensional constraint parameters, reference parameters, and user variables in the current drawing.

PARAMETERSCLOSE

Closes the Parameters Manager palette.

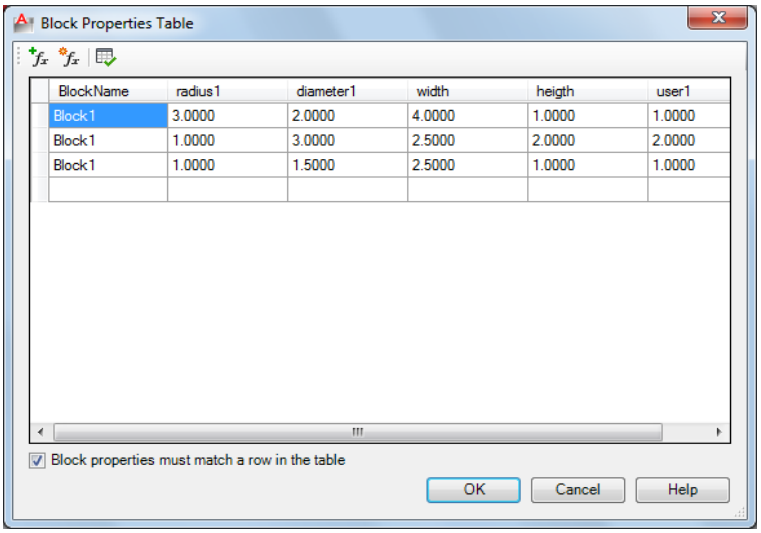
PARAMETERSSTATUS

Indicates whether the Parameters Manager is displayed or hidden.

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.

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.

Quick Reference

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, or converts associative dimensions to dimensional constraints.

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 865.

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:

- Apply or Remove Geometric Constraints on page 872

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.

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.

Quick Reference

BESETTINGS

Displays the Block Editor Settings dialog box.

BCONSTATUSMODE

Turns the constraint display status on and off and controls the shading of objects based on their constraint level.

Add Action Parameters to Dynamic Blocks

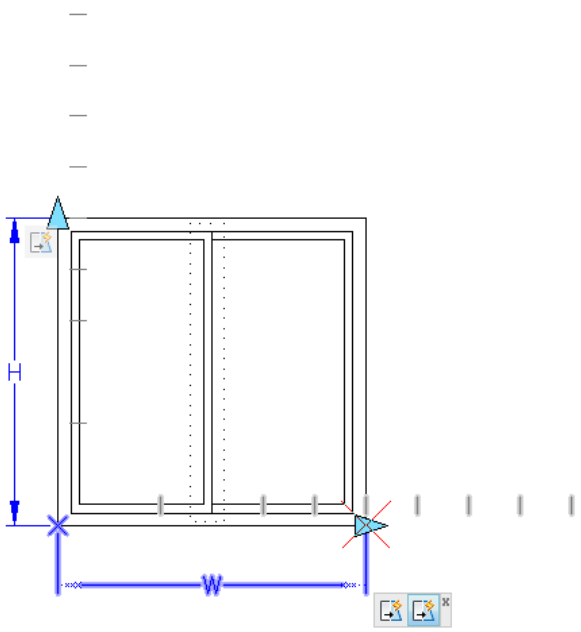
You can add actions and parameters to dynamic blocks to determine how a block should behave when it is inserted into a drawing.

Overview of Actions and Parameters

Parameters, actions, and their relationships (dependencies) are displayed in different ways in the Block Editor. You can specify settings for some of these elements.

Actions

An action displays its name and icon (a lightning bolt) in the Block Editor. Actions are grouped into bars, which display all actions associated with a parameter.



Hover the cursor over an action icon to display

- The associated parameter
- The associated selection set
- Additional objects that belong to the action

NOTE In order to use the BASSOCIATE command, you must first set BACTIONBARMODE to 0.

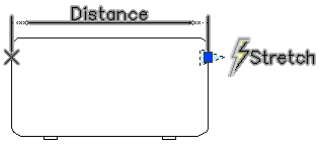
Custom Tools

You can create custom block authoring tools, which you access from the block authoring palettes. 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.

Dependencies

When you select a parameter, grip, or action in the Block Editor, its associated objects 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 settings for parameters and actions 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, set the desired options.
4. Click OK.

To specify the text color for actions in the Block Editor

1. At the Command prompt, enter **bactionbarmode**.
2. Enter **0** to view legacy action objects.

3 At the Command prompt, enter **bactioncolor**.

4 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**.

5 Press Enter.

6 At the Command prompt, enter **regen** to regenerate the drawing.

Quick Reference

BACTIONBAR

Displays or hides action bars for a selection set of parameter objects.

BEDIT

Opens the block definition in the Block Editor.

BESETTINGS

Displays the Block Editor Settings dialog box.

BLOCK

Creates a block definition from selected objects.

REGEN

Regenerates the entire drawing from the current viewport.

BACTIONBARMODE

Indicates whether the action bars or the legacy action objects are displayed in the Block Editor.

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.

Add Parameters to Dynamic Blocks

Grips

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.

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.

See also:

- Specify Value Sets for Dynamic Blocks on page 1008
- Specify Properties for Dynamic Blocks on page 1003
- Extract Data from Block Attributes on page 946
- Extract Block Attribute Data (Advanced) on page 947

Quick Reference

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.

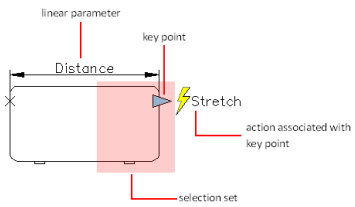
Add Actions to Dynamic Blocks

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.

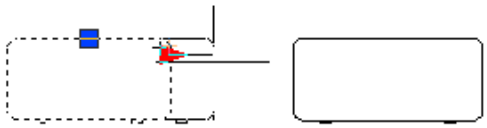
Overview of Actions

In general, you associate an action with a parameter and the following:

- **Key point**. The point on a parameter that drives the action.
- **Selection set**. The geometry that will be affected by the action.



When you move the grip in the example above, only the geometry in the selection set is stretched.



Quick Reference

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.

PROPERTIES

Controls properties of existing objects.

Move Actions

Move selected objects a specified distance and angle.

Apply a Move Action to a Point Parameter

A move action associated with a point parameter moves all objects in a selection set in any direction. The point parameter is the location of the grip for the move action in the block reference.

Apply a Move Action to a Linear Parameter

A move action associated with a linear parameter moves all objects in a selection set only in the direction of the linear parameter. The parameter *key point* is the location of the grip in the block reference.

Apply a Move Action to a Polar Parameter

A move action associated with a polar parameter moves all objects in a selection set in any direction. In the block reference, the parameter *key point* is

- The location of the move grip
- The offset distance from the selected objects

Applying a move action to a polar parameter gives the same result as applying a move action to a point parameter.

Quick Reference

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.

PROPERTIES

Controls properties of existing objects.

Scale Actions

Scale selected objects relative to the base point specified by the action.

Apply a Scale Action to a Linear Parameter

A scale action applied to a linear parameter scales the selected objects in the direction of the parameter.

Apply a Scale Action to a Polar Parameter

A scale action applied to a polar parameter scales the selected objects by the specified distance and angle. While the resulting geometry behaves in the same way as a scale action applied to a linear parameter, the way the grips are manipulated is different in the Properties palette.

Apply a Scale Action to an XY Parameter

A scale action applied to an XY parameter scales the selected objects only in the X and Y direction specified by the parameter. While the resulting geometry behaves in the same way as a scale action applied to a linear parameter, the way the grips are manipulated is different in the Properties palette.

Quick Reference

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.

PROPERTIES

Controls properties of existing objects.

Stretch Actions

Move and stretch objects a specified distance in a specified direction relative to the base point specified by the action.

Apply a Stretch Action to a Point Parameter

A stretch action applied to a point parameter moves and stretches selected objects a specified distance in any direction.

Apply a Stretch Action to a Linear or Polar Parameter

A stretch action applied to a linear parameter moves and stretches selected objects a specified distance in the direction of the parameter.

Apply a Stretch Action to an XY Parameter

A stretch action applied to an XY parameter stretches the selected objects only in the X and Y direction specified by the parameter.

Quick Reference

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.

PROPERTIES

Controls properties of existing objects.

Polar Stretch Actions

Rotate, move, and stretch objects a specified angle and distance.

Apply a Polar Stretch Action to a Polar Parameter

A polar stretch action can only be applied to a polar parameter. Like the stretch action, selected objects stretch when this action is applied. However, a polar stretch action also allows you to stretch the objects at any angle.

Quick Reference

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.

PROPERTIES

Controls properties of existing objects.

Rotate Actions

Rotate selected objects.

A rotate action is always associated with a rotate parameter.

Dependent and Independent Base Points

When you apply a rotate action, you can choose to make the base point of the action dependent or independent of the parameter's base point.

Quick Reference

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.

PROPERTIES

Controls properties of existing objects.

Flip Actions

Mirror selected objects in a dynamic block reference about a specified reflection line.

A flip action is always associated with a flip parameter.

Quick Reference

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.

PROPERTIES

Controls properties of existing objects.

Array Actions

Copy and array objects in a rectangular pattern.

Apply an Array Action to a Linear Parameter

An array action applied to a linear parameter copies and arrays selected objects in a rectangular pattern along the parameter only.

Apply an Array Action to a Polar Parameter

An array action applied to a polar parameter copies and arrays selected objects in a rectangular pattern in any direction and angle.

Apply an Array Action to an XY Parameter

An array action applied to an XY parameter copies and arrays selected objects in a rectangular pattern only in the X and Y direction specified by the parameter.

Quick Reference

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.

PROPERTIES

Controls properties of existing objects.

Lookup Actions

Lookup tables associate parameter values for the dynamic block reference with other specified data (for example, a model or part number).

You can extract data from block references just as you would extract block attribute data.

When the parameter values in a dynamic block reference match a row of input property values in the lookup table, the corresponding lookup property values are assigned to the block reference. These lookup properties and values are displayed in the Properties palette under Custom.

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)

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 980.

Add Properties and Values to a Lookup Table

The Property Lookup Table consists of input properties and lookup properties.

- **Input properties.** Parameters. Each parameter can correspond to one column (except for lookup, alignment, and base point parameters). The parameter label is used as the property name.
- **Lookup properties.** Lookup parameters. Each lookup parameter can correspond to one column. The lookup parameter label is used as the property name.

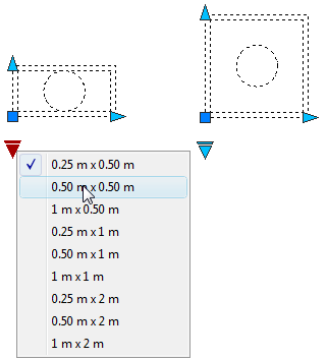
To add values to the cells in each column, click a cell and enter a value. If you defined a value set for an input property (parameter) in the table, a drop-down list of available values will display. Make sure to follow the guidelines in Specify Values for Lookup Tables on page 998.

Custom Values

In a lookup table that does not use value sets, parameter values that do not match a row of input property values from the lookup table are assigned the lookup property value that corresponds to <Unmatched>. The default value is *Custom*.

Enable Reverse Lookup

A lookup property with reverse lookup enabled adds a lookup grip to the dynamic block reference. When you click this grip, a drop-down list of that column's lookup values is displayed. The corresponding input property values of the selected list item are assigned to the block reference.



To enable reverse lookup for a lookup property, each row in the lookup table must be unique. To check for errors and empty cells, in the Property Lookup Table dialog box, click the Audit button. You can also use the shortcut menu options to insert, delete, or reorder rows.

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.

See also:

- Extract Data from Block Attributes on page 946
- Extract Block Attribute Data (Advanced) on page 947

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.

To edit a lookup table while the action bars are displayed in the Block Editor

1 Hover the cursor over the lookup action icon.

2 Right-click and select Display Lookup Table.

To display and add 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 or Add Lookup 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.

6 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.

7 Click OK.

To allow or disallow 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, do one of the following:

- Click Read Only, and then select Allow Reverse Lookup from the list.
- Click Allow Reverse Lookup, and then select Read Only from the list.

NOTE You can only select Allow Reverse Lookup if you have entered data in the column.

4 Click OK.

Quick Reference

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

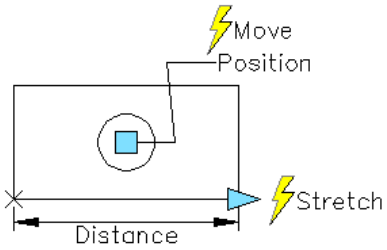
Adds a parameter with grips to a dynamic block definition.

Use Chained Actions

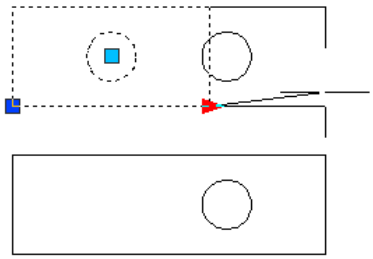
Point, linear, polar, XY, and rotation parameters have a property called *Chain Actions*. This property affects the parameter behavior if the parameter is part of an action's selection set.

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.

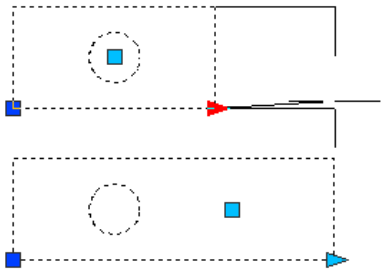
The following example shows a block definition in the Block Editor. The point parameter (labeled Position) is included in the stretch action's selection set.



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.

Quick Reference

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 Properties for Dynamic Blocks

Control the way parameters are displayed using the Properties palette.

Custom Properties

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.

Other parameter properties, such as Value Set properties and Chain Actions, define how the block reference will function in a drawing.

Assign Labels to Parameters

When you select the dynamic block reference in a drawing, parameter labels are assigned to custom properties in the Properties palette. 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.

It is good practice to specify unique parameter labels within the block.

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.

List of Properties for Action Parameters

The following properties are available to customize the way your action parameters behave.

Property name	Description	Associated parameter
■ Angle Name ■ Distance Name ■ Flip Name ■ Hor Distance Name ■ Lookup Name ■ Position Name ■ Ver Distance Name ■ Visibility Name	Defines a label for the parameter. Displayed in the drawing space of the Block Editor and the custom properties of the block reference when the grip is selected.	■ Point ■ Linear ■ Polar ■ XY ■ Rotation ■ Flip ■ Visibility ■ Lookup

Property name	Description	Associated parameter
■ Angle Description ■ Distance Description ■ Flip Description ■ Hor Distance Description ■ Position Description ■ Ver Distance Description ■ Visibility Description ■ Lookup Description	Defines a description that will further explain the purpose of the parameter. The text string will display in the tooltip when the mouse pointer hovers over the position name.	■ Point ■ Linear ■ Polar ■ XY ■ Rotation ■ Flip ■ Visibility ■ Lookup
■ Base State Name ■ Flipped State Name	In a Flip parameter, displays the label that will be displayed when the specified object is displayed as it was drawn (Base State) or as manipulated (Flipped State). Displayed in the custom properties of the block reference when the grip is selected.	Flip
Base Type	In a Rotate action, specifies whether the base point for the rotation is dependent on the base point of the parameter or an independent base point that you specify in the block definition.	Rotate
Distance Type	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.	■ Move ■ Stretch
Show Properties	Specifies whether custom properties will be displayed in the Properties palette for the selected block reference.	■ Point ■ Linear ■ Polar

Property name	Description	Associated parameter
		■ XY ■ Rotate ■ Flip ■ Visibility ■ Lookup

See also:

- Define User Parameters in Dynamic Blocks on page 978
- Specify Value Sets for Dynamic Blocks on page 1008
- Use Chained Actions on page 1001
- Specify Grips for Dynamic Blocks on page 1011
- Extract Data from Block Attributes on page 946

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.

To specify that dynamic block reference can be exploded

1 Open a block definition in the Block Editor

2 In the Block Editor, make sure nothing is selected.

3 In the Properties palette, under Block, click Allow Exploding.

4 On the drop-down list, select Yes or No.

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 prevent non-uniform scaling in dynamic block reference

1 Open a block definition in the Block Editor

2 In the Block Editor, make sure nothing is selected.

3 In the Properties palette, under Block, click Scale Uniformly.

4 On the drop-down list, select Yes.

5 Click Block Editor tab ➤ Open/Save panel ➤ Save Block. 

6 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Quick Reference

BEDIT

Opens the block definition in the Block Editor.

BPARAMETER

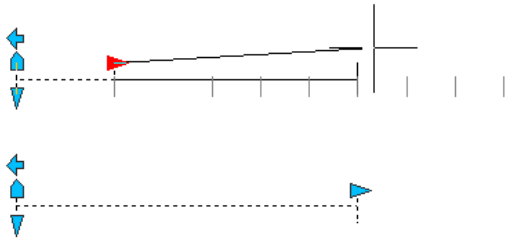
Adds a parameter with grips to a dynamic block definition.

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.

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 add or remove a value set for a 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 parameter.
4. In the Properties palette, under Value Set, click one of the following:
   - Ang Type

- Dist Type
- Hor Type
- Ver Type

5 On the drop-down list, select List.

6 In the Properties palette, under Value Set, click one of the following:
- Ang Value List
- Dist Value List
- Hor Value List
- Ver Value List

7 Click the [...] button.

8 In the Add Distance Value/Angle dialog box, enter one value or two or more values separated by commas, or to delete a value, select it from the list.

9 Click Add or Delete.

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.

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.

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.

Quick Reference

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 Distance and Angle Values

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 990
- Stretch on page 993
- Polar Stretch on page 994

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.

Quick Reference

PROPERTIES

Controls properties of existing objects.

Specify Grips for Dynamic Blocks

Parameter Type	Grip Type	Actions You Can Associate with a Parameter
Point	 Standard	Move, Stretch

Parameter Type	Grip Type	Actions You Can Associate with a Parameter
Linear	 Linear	Move, Scale, Stretch, Array
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

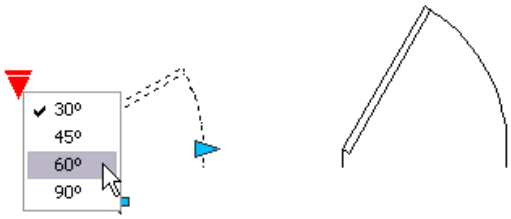
When you add a parameter to a dynamic block definition, custom grips associated with key points of the parameter are automatically added to the block.

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 on page 1017 or a lookup table on page 997, 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.

NOTE Grips are not displayed for key points that are not associated with an action.

The type of parameter you add to the dynamic block definition determines the type of grips that are added to the block..

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

Grip Type		How the Grip Can Be Manipulated in a Drawing	Associated Parameters
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.

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. 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.

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.

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.

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

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.

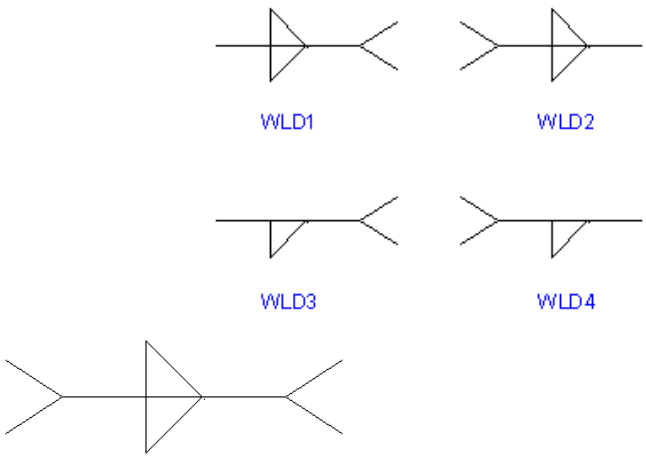
GRIPTIPS

Controls the display of grip tips when the cursor hovers over grips on dynamic blocks and custom objects that support grip tips.

Control the Visibility of Objects in a Block

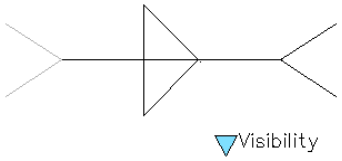
Using visibility states, you can create a block with different graphical representations.

For example, you have the following four different weld symbols. Using visibility states, you can combine these weld symbols into a single dynamic block.



After you combine the geometry in the Block Editor, you add a visibility parameter.

You can then create and name a different visibility state for each weld symbol (for example, WLD1, WLD2, WLD3, and WLD4). 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 Block Editor contextual tab displays the name of the current visibility 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. Use the Visibility Mode button (BVMODE) to determine whether geometry is displayed.

To create a new 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, click New.

4 In the New Visibility State dialog box, enter a name for the new visibility state.

5 Do one of the following:

- Click Hide All Existing Objects in New State
- Click Show All Existing Objects in New State
- Click Leave Visibility of Existing Objects Unchanged in New State

6 Click OK.

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.

To rename or 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 Do one of the following:

- Click Rename and enter a new name for the visibility state
- Click Delete

5 Click OK.

To make objects visible or invisible in the current visibility state

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. Do one of the following:
   - Click Block Editor tab ➤ Visibility panel ➤ Make Visible. 
   - Click Block Editor tab ➤ Visibility panel ➤ Make Invisible. 
3. In the drawing area, select the objects. Press Enter.
4. At the Command prompt, enter **current**. Press Enter.

To make objects visible or invisible 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 and click one of the following:
   - Object Visibility ➤ Show For All States
   - Object Visibility ➤ Hide For All States

To show or hide invisible geometry for visibility states

1. Click Insert tab ➤ Block panel ➤ Block Editor. 
2. 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 desired order.
6. Click OK.

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.

Quick Reference

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.

BVMODE

Controls how objects that are made invisible for the current visibility state are displayed in the Block Editor.

Part 8: Work with 3D Models

Create 3D Models

24

Use 3D models to help you visualize and test your designs.

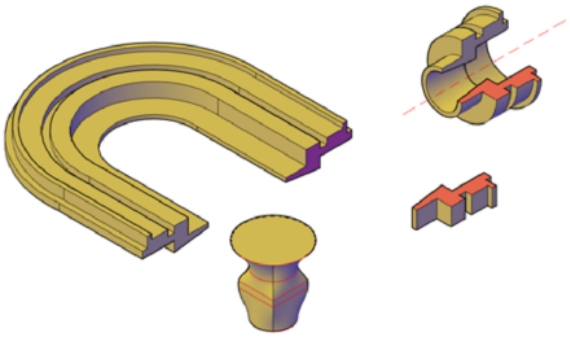
Overview of 3D Modeling

AutoCAD 3D modeling allows you to create drawings using solid, surface, and mesh objects.

Solid, surface, and mesh objects offer different functionality, that, when used together, offer a powerful suite of 3D modeling tools. For example, you can convert a primitive solid to a mesh to take advantage of mesh creasing and smoothing. You can then convert the model to a surface to take advantage of associativity and NURBS modeling.

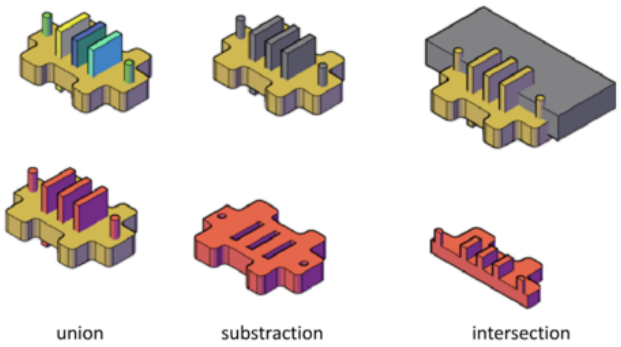
Solid Modeling

A solid model is an enclosed 3D body that has properties such as mass, volume, center of gravity, and moments of inertia.



Start with primitive solids such as cones, boxes, cylinders, and pyramids and modify and recombine them to create new shapes. Or draw a custom polysolid

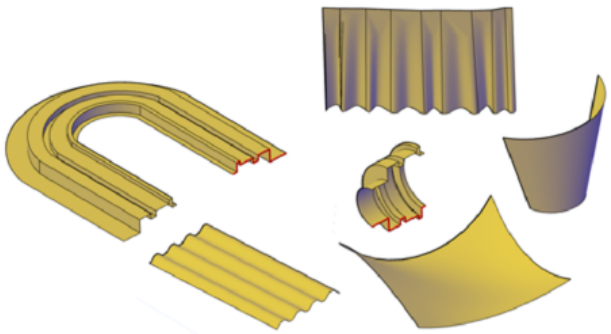
extrusion and use various sweeping operations to create solids from 2D curves and lines.



Surface Modeling

A surface model is a thin shell that does not have mass or volume. AutoCAD offers two types of surfaces: procedural and NURBS. Use prodecural surfaces to take advantage of associative modling, and use NURBS surfaces to take advantage of sculpting with control vertices.

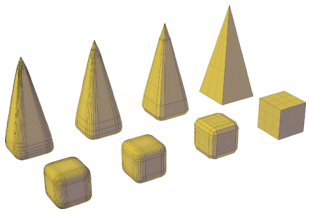
A typical modeling workflow is to create a basic model using mesh, solids, and procedural surfaces, and then convert them to NURBS surfaces. This allows you to utilize not only the unique tools and primitive shapes offered by solids and meshes, but also the shaping capabilities provided by surfaces - associative modeling and NURBS modeling.



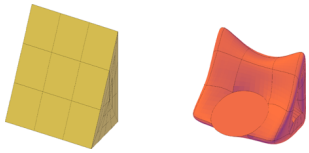
You create surface models using some of the same tools that you use for solid models: sweeping, lofting, extruding, and revolving. You can also create surfaces by blending, patching, offsetting, filleting, and extending other surfaces.

Mesh Modeling

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 . You can 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 614
- Control the User Coordinate System (UCS) on page 594
- Use the Dynamic UCS with Solid Models on page 601

Quick Reference

Commands

BOX

Creates a 3D solid box.

CONE

Creates a 3D solid cone.

CONVTOMESH

Converts 3D objects such as polygon meshes, surfaces, and solids to mesh objects.

CONVTONURBS

Converts 3D solids and surfaces into NURBS surfaces.

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

Creates a 3D solid or surface by extending the dimensions of an object.

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.

PLANESURF

Creates a planar surface.

PYRAMID

Creates a 3D solid pyramid.

REVOLVE

Creates a 3D solid or surface by sweeping an object around an axis.

SPHERE

Creates a 3D solid sphere.

SURFBLEND

Creates a continuous blend surface between two existing surfaces.

SURFNETWORK

Creates a surface in the space between several curves in the U and V directions (including surface and solid edge subobjects).

SURFOFFSET

Creates a parallel surface a specified distance from the original surface.

SURFPATCH

Creates a new surface by fitting a cap over a surface edge that forms a closed loop.

SWEEP

Creates a 3D solid or surface by sweeping a 2D or 3D object or subobject along a path.

TORUS

Creates a donut-shaped 3D solid.

UCS

Sets the origin and orientation of the current UCS.

WEDGE

Creates a 3D solid wedge.

System Variables

DELOBJ

Controls whether geometry used to create other 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.

SUBOBJSELECTIONMODE

Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

SURFACEASSOCIATIVITY

Controls whether surfaces maintain a relationship with the objects from which they were created.

SURFACEMODELINGMODE

Controls whether surfaces are created as procedural surfaces or NURBS surfaces.

Create Solids and Surfaces from Lines and Curves

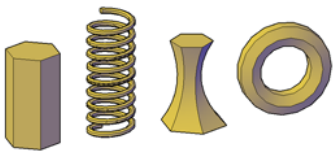
Use lines and curves to extrude, sweep, loft, and revolve 3D solids, surfaces, and NURBS surfaces.

Overview of Creating Solids and Surfaces

Understand the differences between creating solids and surfaces with the EXTRUDE, SWEEP, LOFT, and REVOLVE commands.

See also:

- Create a Solid or Surface by Extruding on page 1034
- Create a Solid or Surface by Sweeping on page 1038
- Create a Solid or Surface by Lofting on page 1042
- Create a Solid or Surface by Revolving on page 1046



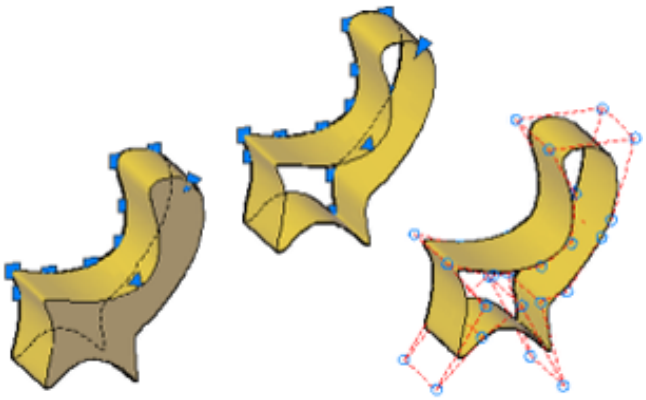
Create Surfaces or Solids With the Same Commands

When you extrude, sweep, loft, and revolve curves, you can create both solids and surfaces. Open curves always create surfaces, but closed curves can create either solids or surfaces depending on the situation.

If you select a **closed** curve and click EXTRUDE, SWEEP, LOFT, and REVOLVE on the ribbon, you create:

- A **solid** if the Mode option is set to Solid.
- A **surface** if the Mode option is set to Surface.
  - A **procedural surface** if the SURFACEMODELINGMODE system variable is set to 0.
  - A **NURBS surface** if the SURFACEMODELINGMODE system variable is set to 1.
  - An **associative surface** if the SURFACEASSOCIATIVITY system variable is on.

In this illustration, the same profile creates a solid (left), a procedural surface (middle), and a NURBS surface (right).



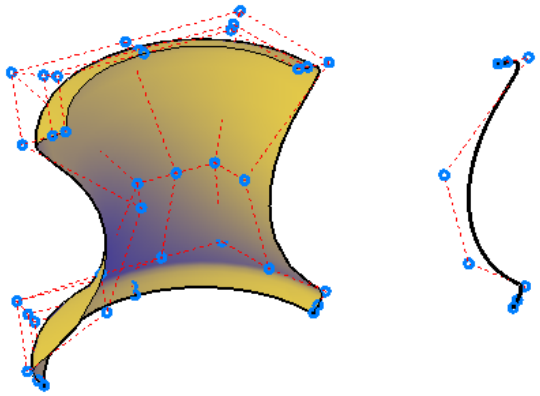
Geometry That Can Be Used As Profiles and Guide Curves

The curves that you use as profile and guide curves when you extrude, sweep, loft, and revolve can be:

- Open or closed
- Planar or non-planar
- Solid and surface edge subobjects
- A single object (to extrude multiple lines, convert them to a single object with the JOIN command)
- A single region (to extrude multiple regions, convert them to a single object with the REGION command)

Example: Use Splines to Create 3D NURBS Surfaces

Splines are one of the many 2D object types that can be lofted, extruded, swept, and revolved to create NURBS surfaces. Other 2D objects that can be used include lines, polylines, arcs, and circles. Splines, however, are the only 2D object customized to create NURBS surfaces. Because they allow you to adjust tolerance, degree, and tangency, they are better suited than other types of 2D profiles (line, plines, circles) for surface modeling.



Many of the same commands used with NURBS surfaces, can also be used with CV splines. These include:

- CVADD
- CVREMOVE
- CVREBUILD
- CVSHOW
- CVHIDE

For more information, see Create Solids and Surfaces from Lines and Curves on page 1030.

Create Associative Surfaces

Surfaces can be associative while solids cannot. If surface associativity is on when a surface is created, it maintains a relationship with the curve from which it is was generated (even if the curve is the subobject of another solid or surface). If the curve is reshaped, the surface profile automatically updates. See Create Associative Surfaces on page 1103.

NOTE To modify a surface that is associative, you must modify the generating curve and not the surface itself. If you reshape the surface, its link to the generating curve will be broken and the surface will lose associativity and become a generic surface.

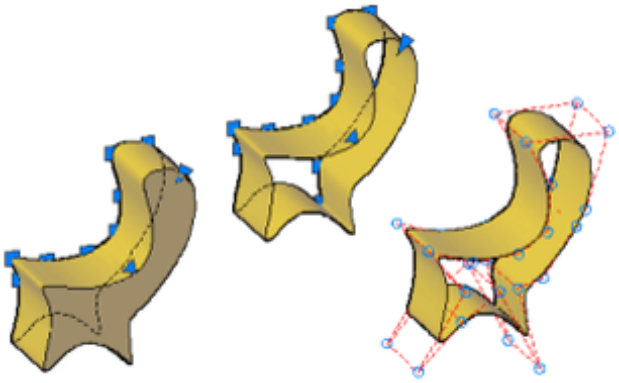
Deleting the Curves that Generate the Solid or Surface

The DELOBJ system variable controls whether the curves that generate an object are automatically deleted after the solid or surface is created. However, if surface associativity is on, the DELOBJ setting is ignored and the generating curves are not deleted.

Create a Solid or Surface by Extruding

Create a 3D solid or surface by stretching curves into 3D space.

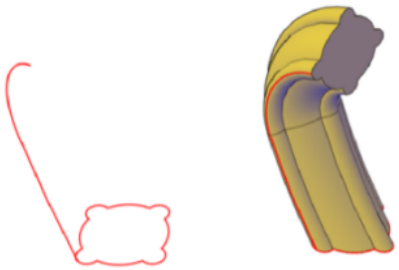
The EXTRUDE command creates a solid or surface that extends the shape of a curve. Open curves create surfaces and closed curves create solids or surfaces. See Overview of Creating Solids and Surfaces on page 1030.



Options for Extrusion

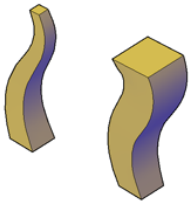
When you extrude objects, you can specify any of the following options:

- **Mode.** Sets whether the extrude creates a surface or a solid.
- **Specify a path for 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 object 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.

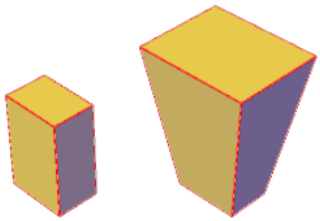


Extruding is different from sweeping. When you extrude a profile along a path, the profile follows the shape of the path, even if the path does not intersect the profile. Sweeping usually provides greater control and better results.

- **Taper angle.** Tapering the extrusion is useful for defining part that require a specific taper angle, such as a mold used to create metal products in a foundry.



- **Direction.** With the Direction option, you can specify two points to set the length and direction of the extrusion.



- **Expression.** Enter a mathematical expression to constrain the height of the extrusion. See Create Geometric Relationships between Associative Surfaces on page 1106.

To extrude a NURBS surface

1 Click Surface tab ➤ Create panel ➤ NURBS Creation. 

2 Click Home tab ➤ Surface panel ➤ Extrude. 

3 Select the objects or edge subobjects to extrude.

4 Specify the height.

After the extrusion, the original objects are deleted or retained, depending on the setting of the DELOBJ system variable.

To extrude a solid

1 Click Solid tab ➤ Create panel ➤ Extrude. 

2 Select the objects or edge subobjects 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.

To extrude a procedural surface along a path

1 Click Surface tab ➤ Create panel ➤ Extrude.

2 Select the objects or edge subobjects to extrude.

3 At the Command prompt, enter **p** (Path).

4 Select the object or edge subobject to use as the path.

After the extrusion, the original objects are deleted or retained, depending on the setting of the DELOBJ system variable.

Quick Reference

Commands

3DOSNAP

Sets the object snap modes for 3D objects.

BREP

Removes the history from 3D solids and composite solids, and associativity from surfaces.

EXTRUDE

Creates a 3D solid or surface by extending the dimensions of an object.

JOIN

Joins the endpoints of linear and curved objects to create a single object.

LIST

Displays property data for selected objects.

MEASUREGEOM

Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

REGION

Converts an object that encloses an area into a region object.

System Variables

3DOSMODE

Controls the settings for the 3D object snaps.

DELOBJ

Controls whether geometry used to create other objects is retained or deleted.

SOLIDHIST

Controls whether new composite solids retain a history of their original components.

SUBOBJSELECTIONMODE

Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

SURFACEASSOCIATIVITY

Controls whether surfaces maintain a relationship with the objects from which they were created.

SURFACEASSOCIATIVITYDRAG

Sets the dragging preview behavior of associative surfaces.

SURFACEMODELINGMODE

Controls whether surfaces are created as procedural surfaces or NURBS surfaces.

Create a Solid or Surface by Sweeping

Create a 3D solid or surface by sweeping a profile along a path.

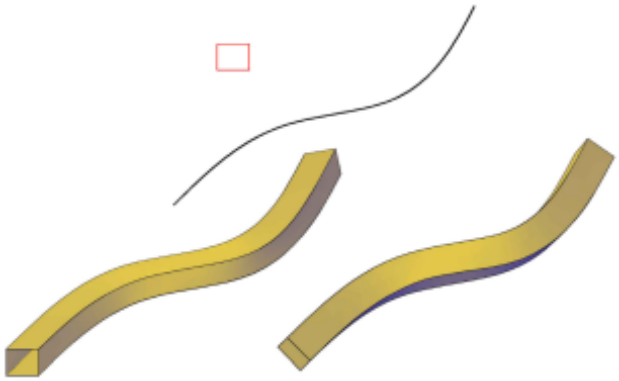
The SWEEP command creates a solid or surface by extending a profile shape (the object that is swept) along a specified path. When you sweep a profile along a path, the profile is moved and aligned normal (perpendicular) to the path. Open profiles create surfaces and closed curves create solids or surfaces. See Overview of Creating Solids and Surfaces on page 1030.

You can sweep more than one profile object along a path.

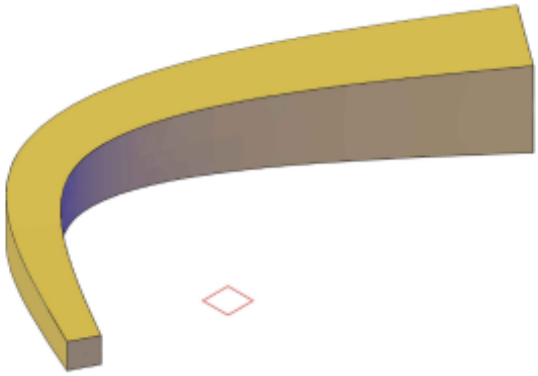
Options for Sweeping

When you extrude objects, you can specify any of the following options:

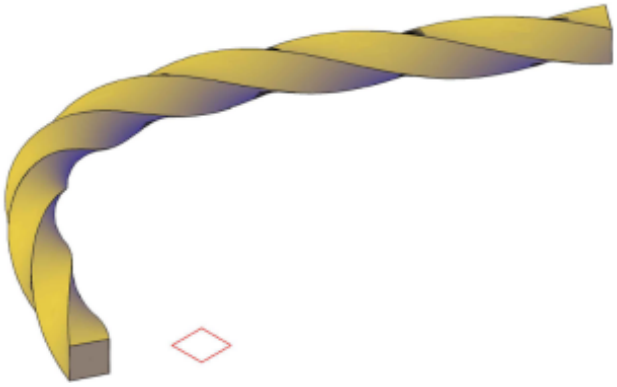
- **Mode.** Sets whether the sweep creates a surface or a solid.
- **Alignment.** If the profile is not on the same plane as the sweep path, specify how the profile aligns with the sweep path.



- **Base Point.** Specify a base point on the profile to sweep along the profile.
- **Scale.** Specify a value that will change the size of the object from the beginning of the sweep to the end. Enter a mathematical expression to constrain the object scaling. See Create Geometric Relationships between Associative Surfaces on page 1106.



- **Twist.** By entering a twist angle, the object rotates along the length of the profile. Enter a mathematical expression to constrain the object 's twist angle. See Create Geometric Relationships between Associative Surfaces on page 1106.



See also:

- Create Associative Surfaces on page 1103
- Create Geometric Relationships between Associative Surfaces on page 1106

- Draw Splines on page 723
- Modify Splines on page 853

To create a solid by sweeping an object along a path

1. Click Solid tab ➤ Create panel ➤ Sweep. 
2. Select objects to sweep.
3. Press Enter.
4. Select an object or edge subobject as a sweep path.

   After the sweep, the original objects are deleted or retained, depending on the setting of the DELOBJ system variable.

To create a surface by sweeping an object along a path

1. Click Surface tab ➤ Create panel ➤ Sweep. 
2. Select objects to sweep.
3. Press Enter.
4. Select an object or edge subobject as a sweep path.

   After the sweep, the original objects are deleted or retained, depending on the setting of the DELOBJ system variable.

Quick Reference

Commands

3DOSNAP

Sets the object snap modes for 3D objects.

BREP

Removes the history from 3D solids and composite solids, and associativity from surfaces.

JOIN

Joins the endpoints of linear and curved objects to create a single object.

LIST

Displays property data for selected objects.

MEASUREGEOM

Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

REGION

Converts an object that encloses an area into a region object.

SWEEP

Creates a 3D solid or surface by sweeping a 2D or 3D object or subobject along a path.

System Variables

3DOSMODE

Controls the settings for the 3D object snaps.

DELOBJ

Controls whether geometry used to create other objects is retained or deleted.

SOLIDHIST

Controls whether new composite solids retain a history of their original components.

SUBOBJSELECTIONMODE

Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

SURFACEASSOCIATIVITY

Controls whether surfaces maintain a relationship with the objects from which they were created.

SURFACEASSOCIATIVITYDRAG

Sets the dragging preview behavior of associative surfaces.

SURFACEMODELINGMODE

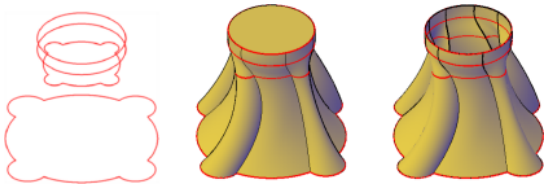
Controls whether surfaces are created as procedural surfaces or NURBS surfaces.

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.

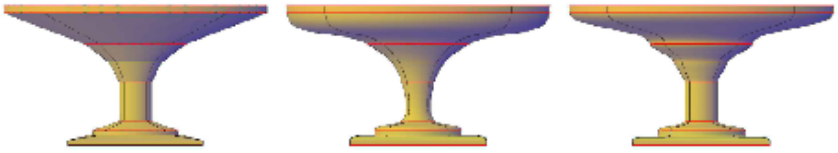
The cross-section profiles define the shape of the resulting solid object.

Cross-section profiles can be open or closed curves. Open curves create surfaces and closed curves create solids or surfaces. See Overview of Creating Solids and Surfaces on page 1030.



Options for Lofting

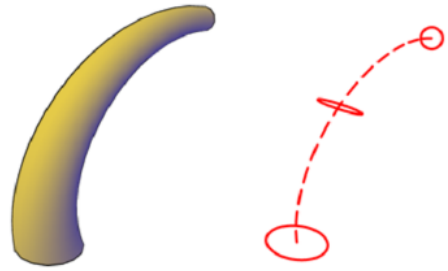
- **Mode.** Sets whether the loft creates a surface or a solid.
- **Cross-section profiles.** Select a series of cross-section profiles to define the shape of the new 3D object.



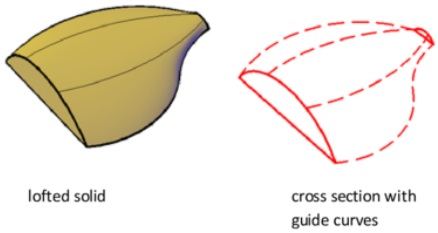
lofted objects with different cross-section settings

As you create a lofted object, you can adjust its shape by specifying how the profile passes through the cross sections (for example, a sharp or smooth curve). You can also modify the settings later in the Properties palette. For more information, see Modify the Properties of 3D Solids, Surfaces, and Meshes on page 1213.

- **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

See also:

- Create Associative Surfaces on page 1103
- Create Geometric Relationships between Associative Surfaces on page 1106
- Draw Splines on page 723
- Modify Splines on page 853

To create a NURBS surface by lofting

1. Click Surface tab ➤ Create panel ➤ NURBS Creation. 
2. Click Surface tab ➤ Create panel ➤ Loft. 
3. 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.)

After the loft operation, the original objects are deleted or retained, depending on the setting of the DELOBJ system variable.

To create a solid by lofting through a set of cross-section profiles

1. Click Solid tab ➤ Create 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.

Quick Reference

Commands

3DOSNAP

Sets the object snap modes for 3D objects.

BREP

Removes the history from 3D solids and composite solids, and associativity from surfaces.

JOIN

Joins the endpoints of linear and curved objects to create a single object.

LIST

Displays property data for selected objects.

MEASUREGEOM

Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

LOFT

Creates a 3D solid or surface in the space between several cross sections.

REGION

Converts an object that encloses an area into a region object.

System Variables

3DOSMODE

Controls the settings for the 3D object snaps.

DELOBJ

Controls whether geometry used to create other objects is retained or deleted.

PREVIEWCREATIONTRANSPARENCY

Controls the transparency of the preview generated while using SURFBLEND, SURFPATCH, SURFFILLET, FILLETEDGE, CHAMFEREDGE, and LOFT.

SOLIDHIST

Controls whether new composite solids retain a history of their original components.

SUBOBJSELECTIONMODE

Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

SURFACEASSOCIATIVITY

Controls whether surfaces maintain a relationship with the objects from which they were created.

SURFACEASSOCIATIVITYDRAG

Sets the dragging preview behavior of associative surfaces.

SURFACEMODELINGMODE

Controls whether surfaces are created as procedural surfaces or NURBS surfaces.

Create a Solid or Surface by Revolving

Create a 3D object by revolving curves about an axis.

When the Mode option is set to Surface, you will create a surface and if Mode is set to Solid you will create a solid regardless of whether the curve is open or closed. See Overview of Creating Solids and Surfaces on page 1030 for more information. When revolving a solid, you can only use a revolve angle of 360 degrees.

Options for Revolving

- **Mode.** Sets whether the revolve creates a surface or a solid.
- **Start Angle.** Specifies an offset for the revolution from the plane of the object being revolved.
- **Reverse.** Changes the direction of the revolve.
- **Expression.** Enter a formula or equation to specify the revolve angle. This option is only available if you are creating associative surfaces. See Create Geometric Relationships between Associative Surfaces on page 1106.

See also:

- Create Associative Surfaces on page 1103
- Draw Splines on page 723
- Modify Splines on page 853

To revolve objects about an axis to create a NURBS surface

1. Click Surface tab ➤ Create panel ➤ Revolve. 
2. Select the objects or edge subobjects 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.

To revolve objects about an axis to create a solid

1. Click Solid tab ➤ Create panel ➤ Revolve. 
2. Select the objects or edge subobject 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. Press Enter. To create a solid the angle must be 360 degrees. If you enter a smaller angle of revolution, a surface is created instead of a solid.

Quick Reference

Commands

3DOSNAP

Sets the object snap modes for 3D objects.

BREP

Removes the history from 3D solids and composite solids, and associativity from surfaces.

JOIN

Joins the endpoints of linear and curved objects to create a single object.

LIST

Displays property data for selected objects.

MEASUREGEOM

Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

REGION

Converts an object that encloses an area into a region object.

REVOLVE

Creates a 3D solid or surface by sweeping an object around an axis.

System Variables

3DOSMODE

Controls the settings for the 3D object snaps.

DELOBJ

Controls whether geometry used to create other objects is retained or deleted.

SOLIDHIST

Controls whether new composite solids retain a history of their original components.

SUBOBJSELECTIONMODE

Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

SURFACEASSOCIATIVITY

Controls whether surfaces maintain a relationship with the objects from which they were created.

SURFACEASSOCIATIVITYDRAG

Sets the dragging preview behavior of associative surfaces.

SURFACEMODELINGMODE

Controls whether surfaces are created as procedural surfaces or NURBS surfaces.

Create Solids

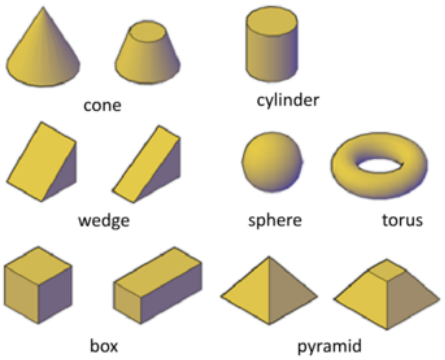
Create 3D solids from primitives or by combining or extending existing objects.

Overview of Creating 3D Solids

3D solid objects often start as one of several basic shapes, or primitives, that you can then modify and recombine. A 3D solid can also be the result of extruding a 2D shape to follow a specified path in 3D space.

About Solid Primitives

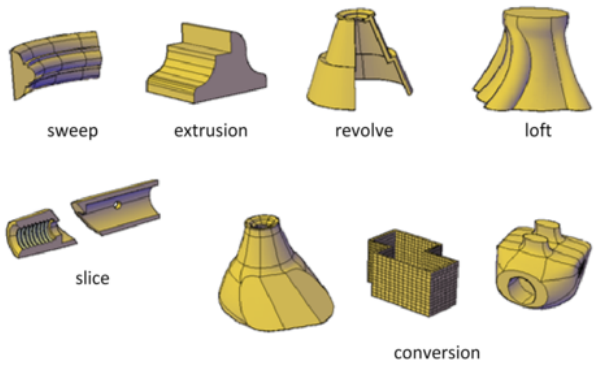
You can create several basic 3D shapes, known as *solid primitives*: boxes, cones, cylinders, spheres, wedges, pyramids, and tori (donuts).



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 from 2D geometry or other 3D 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.
- **Sculpting Surfaces.** Converts and trims a group of surfaces that enclose a watertight area into a solid.
- **Conversion.** Converts mesh objects and planar objects with thickness into solids and surfaces.

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

Creates a 3D solid or surface by extending the dimensions of an object.

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 an object around an axis.

SLICE

Creates new 3D solids and surfaces by slicing, or dividing, existing objects.

SUBTRACT

Combines selected 3D solids or 2D regions by subtraction.

SURFSCULPT

Trims and combines surfaces that bound a watertight area to create a solid.

SWEEP

Creates a 3D solid or surface by sweeping a 2D or 3D object or subobject 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

SURFACEMODELINGMODE

Controls whether surfaces are created as procedural surfaces or NURBS surfaces.

FACETRES

Adjusts the smoothness of shaded and rendered objects and objects with hidden lines removed.

ISOLINES

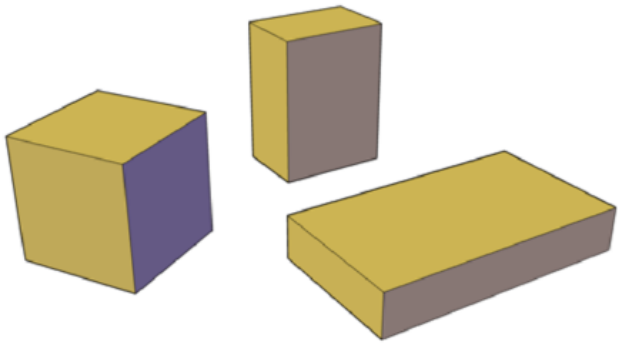
Specifies the number of contour lines displayed on the curved surfaces of 3D solids.

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 (work plane).

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.

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.

Quick Reference

Commands

BOX

Creates a 3D solid box.

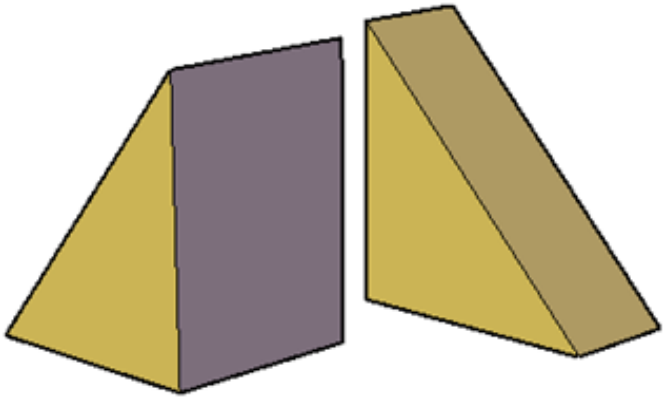
System Variables

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.

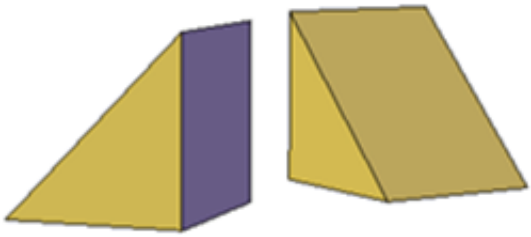


The base of the wedge is drawn parallel to the *XY* plane of the current UCS with the sloped face opposite the first corner. The height of the wedge is parallel to the *Z* axis.

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.

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.

Quick Reference

Commands

WEDGE

Creates a 3D solid wedge.

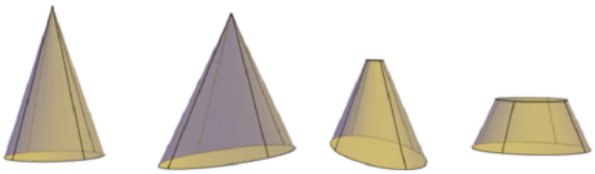
System Variables

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.

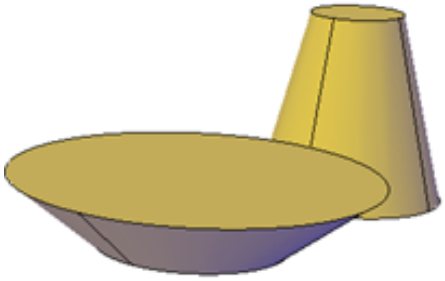


By default, the base of the cone lies on the *XY* plane of the current UCS. The height of the cone is parallel to the *Z* axis.

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 1183

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.

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.

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.

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.

Quick Reference

Commands

CONE

Creates a 3D solid cone.

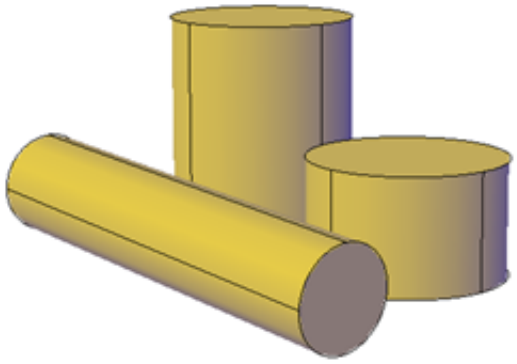
System Variables

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.

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.

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.

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.

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.

Quick Reference

Commands

SPHERE

Creates a 3D solid sphere.

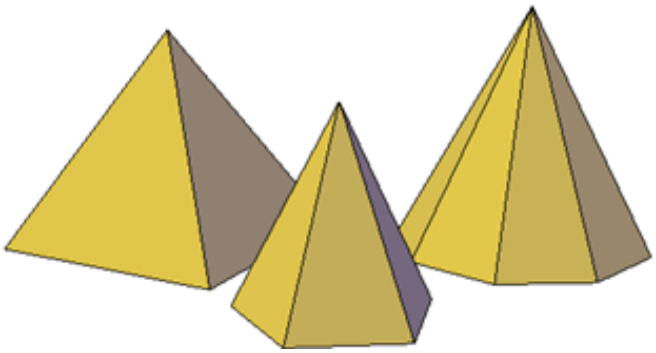
System Variables

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.

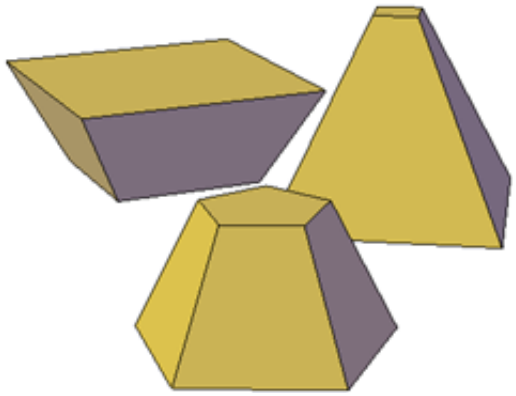


You can create a pyramid that tapers to a point, or create a frustum of a pyramid, which tapers to a planar face.

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.

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.

Quick Reference

Commands

PYRAMID

Creates a 3D solid pyramid.

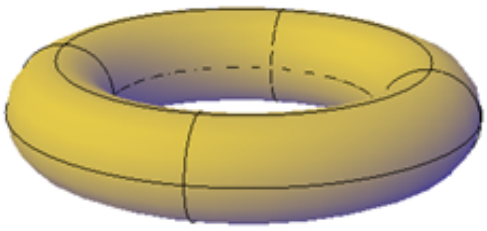
System Variables

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

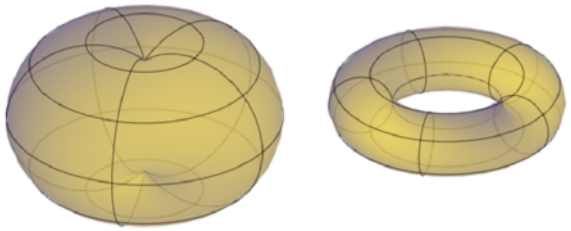
Create a ring-shaped solid that resembles the inner tube of a tire.



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.

Quick Reference

Commands

TORUS

Creates a donut-shaped 3D solid.

System Variables

DRAGVS

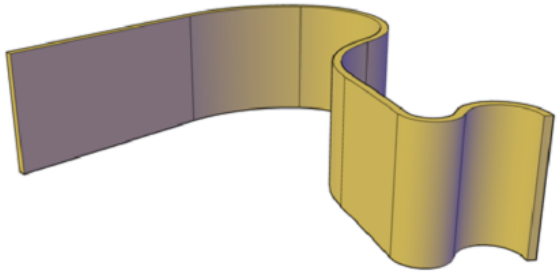
Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

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.

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.

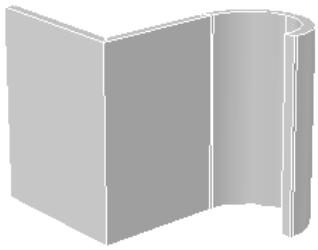
You can also convert objects such as a line, 2D polyline, arc, or circle to a polysolid.

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.

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.

Quick Reference

Commands

POLYSOLID

Creates a 3D wall-like polysolid.

System Variables

DELOBJ

Controls whether geometry used to create other 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.

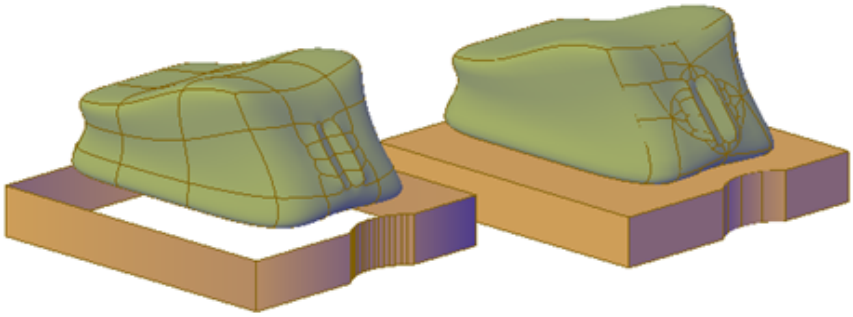
Create 3D Solids from Objects

Convert existing objects to 3D solids.

You can use several methods to convert objects in your drawing to 3D solids:

- Convert surfaces and objects with Thickness to 3D Solids
- Convert a group of surfaces to a 3D solid

- Convert mesh to 3D solids
- Thicken surfaces to convert them 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 different 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.

Convert a Group of Surfaces to a 3D Solid

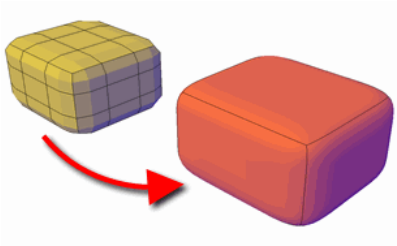
Use the SURFSCULPT command to convert a group of surfaces that enclose a watertight region to a 3D solid.

Convert Mesh to 3D Solids

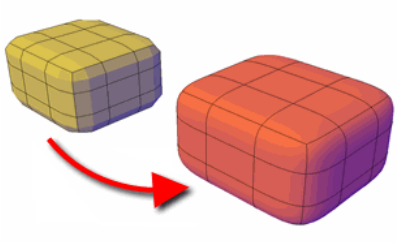
When you convert mesh objects to 3D solids, the shape of the new solid object approximates, but does not exactly duplicate, the original mesh object. You can control the differentiation somewhat by specifying whether the result is smooth or faceted (SMOOTHMESHCONVERT). You can also specify whether the resulting faces are merged (optimized).

For example, if you convert a mesh box to a solid object, following options are available:

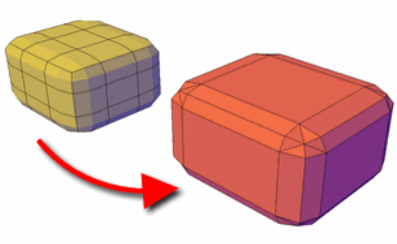
- **Smoothed and optimized.** Coplanar faces are merged into a single face. The overall shape of some faces can change. Edges of faces that are not coplanar are rounded. (SMOOTHMESHCONVERT = 0)



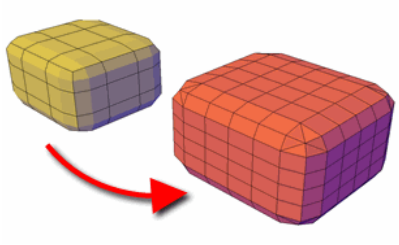
- **Smoothed and not optimized.** Each original mesh face is retained in the converted object. Edges of faces that are not coplanar are rounded. (SMOOTHMESHCONVERT = 1)



- **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. (SMOOTHMESHCONVERT = 2)



- **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. (SMOOTHMESHCONVERT = 3)



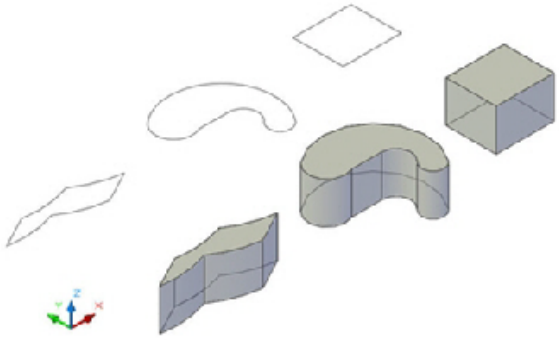
You cannot convert the following types of mesh objects to a 3D solid:

- **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

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.

To convert contiguous surfaces that enclose a volume to a 3D solid object

1. Click Home tab ➤ Solid Editing panel ➤ Sculpt.
2. Select surfaces that enclose a watertight region.
   A solid is created.
3. Click Home tab ➤ Solid Editing panel ➤ Convert to Solid.
4. Select the new compound object.

To convert a mesh object to a 3D solid

1. Click Mesh Modeling tab ➤ Convert Mesh panel ➤ Convert Options drop-down.
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.

Quick Reference

Commands

CONVTOSOLID

Converts 3D meshes and polylines and circles with thickness to 3D solids.

SURFSCULPT

Trims and combines surfaces that bound a watertight area to create a solid.

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 other 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.

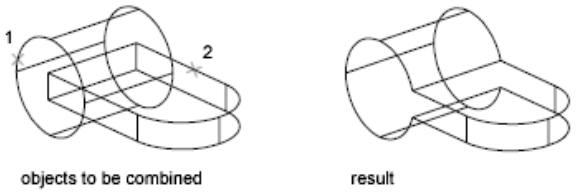
Composite solids are created from two or more solids, surfaces, or regions through any of the following commands: UNION, SUBTRACT, and INTERSECT.

3D solids record a *history* of how they were created. This history allows you to see the original forms that make up composite solids. For more information, see Display Original Forms of Composite Solids on page 1203.

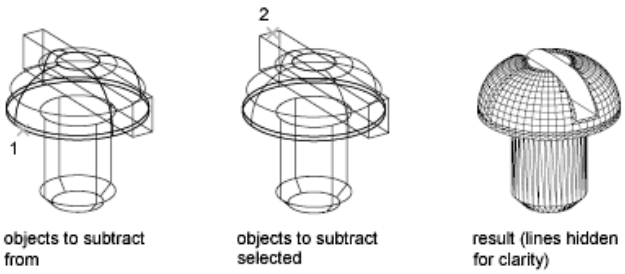
Methods for Creating Composite Objects

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.

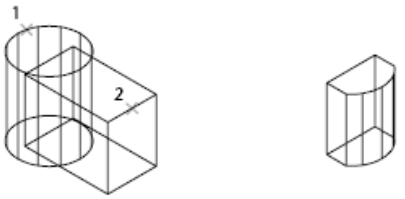


- **Subtract one set of solids from another.**
  With SUBTRACT, you can remove the common area of one set of solids from another. For example, you can use SUBTRACT to add holes to a mechanical part by subtracting cylinders from the object.



- **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.



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 a 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.

You cannot combine solids with mesh objects. However, you can convert them to 3D solids in order to combine them with solids.

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.

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.

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.

Quick Reference

Commands

INTERSECT

Creates a 3D solid, surface, or 2D region from overlapping solids, surfaces, or regions.

SUBTRACT

Combines selected 3D solids 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 whether new composite solids retain a history of their original components.

Create 3D Solids by Slicing

Create new 3D solids by slicing, or dividing, existing objects.

When you use the SLICE command to slice a 3D solid, 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.

See also:

- Create Sections and Drawings from 3D Models on page 1269

To slice solids

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.

To slice solids 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.

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.

Quick Reference

Commands

SLICE

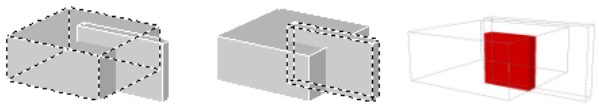
Creates new 3D solids and surfaces by slicing, or dividing, existing objects.

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.

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.

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 Surfaces

Create procedural surfaces and NURBS surfaces from curves or by combining or extending existing surfaces.

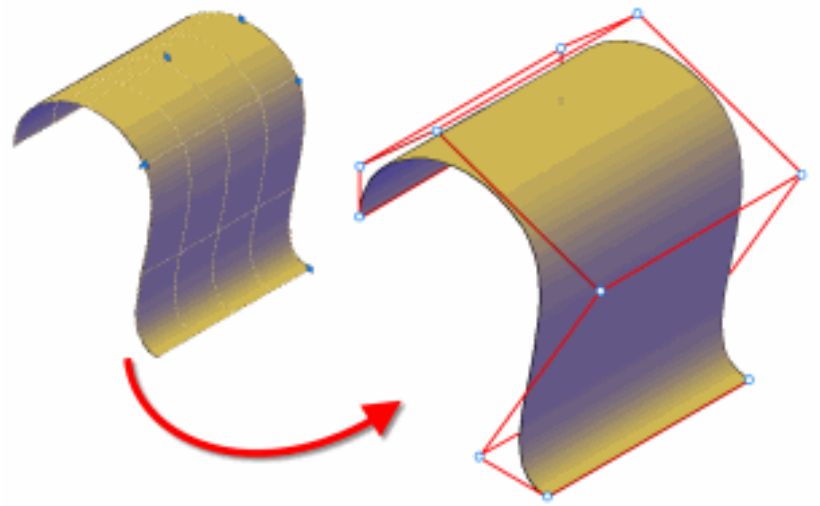
Overview of Creating Surfaces

Surface modeling provides the ability to edit multiple surfaces as an associative group or in a more free-form way.

In addition to 3D solid and mesh objects, AutoCAD offer two type of surfaces: procedural and NURBS.

- **Procedural Surfaces** can be associative, maintaining relationships with other objects so that they can be manipulated as a group.
- **NURBS Surfaces** - are not associative. Instead, they have control vertices that allow you to sculpt shapes in a more natural way.

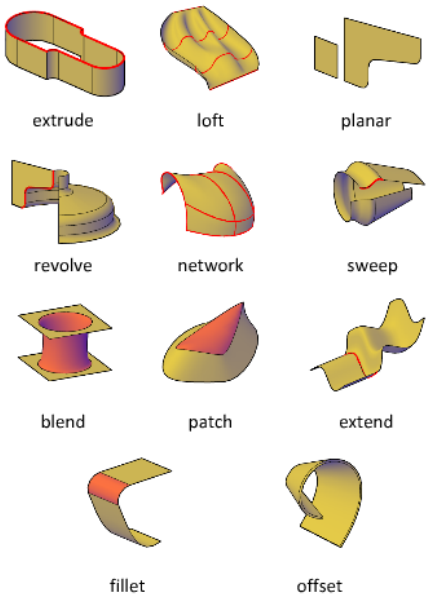
Use procedural surfaces to take advantage of associative modeling, and use NURBS surfaces to take advantage of sculpting with control vertices. The illustration below shows a procedural surface on the left, and a NURBS surface on the right.



Choose a Surface Creation Method

Create procedural and NURBS surfaces using the following methods:

- **Create surfaces from profiles on page 1030.** Create surfaces from profile shapes composed of lines and curves with EXTRUDE, LOFT, PLANESURF, REVOLVE, SURFNETWORK, and SWEEP.
- **Create surfaces from other surfaces on page 1086.** Blend, patch, extend, fillet, and offset surfaces to create new surfaces (SURFBLEND, SURFPATCH, SURFEXTEND, SURFFILLET and SURFOFFSET).
- **Convert objects into procedural surfaces on page 1097.** Convert existing solids (including composite objects), surfaces, and meshes into procedural surfaces (CONVTOSURFACE command).
- **Convert procedural surfaces into NURBS surfaces on page 1097.** Some objects cannot be converted directly to NURBS (for example, mesh objects). In that case, convert the object to a procedural surface and then convert it to a NURBS surface. (CONVTONURBS command).



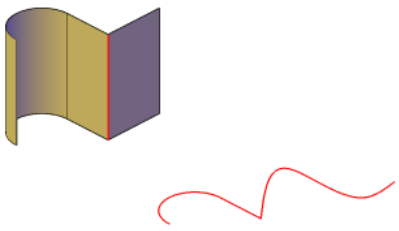
Understand Surface Continuity and Bulge Magnitude

Surface continuity and bulge magnitude are properties that are frequently used when creating surfaces. When you create a new surface, you can specify the continuity and bulge magnitude with special grips.

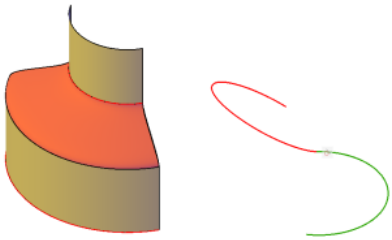
Continuity is a measure of how smoothly two curves or surfaces flow into each other. The type of continuity can be important if you need to export your surfaces to other applications.

Continuity types include the following:

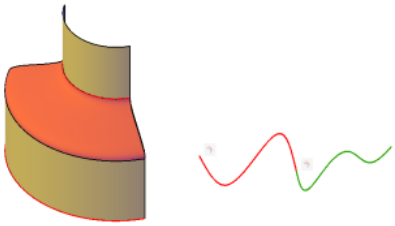
- **G0 (Position).** Measures location only. If the edge of each surface is collinear, the surfaces are positionally continuous (G0) at the edge curves. Note that two surfaces can meet at any angle and still have positional continuity.



- **G1 (Tangency).** Includes both positional and tangential continuity (G0 + G1). With tangentially continuous surfaces, the end tangents match at the common edges. The two surfaces appear to be traveling in the same direction at the join, but they may have very different apparent "speeds" (or rates of change in direction, also called curvature).



- **G2 (Curvature).** Includes positional, tangential, and curvature continuity (G0 + G1+G2). The two surfaces share the same curvature.



Bulge magnitude is a measure of how much surface curves or "bulges" as it flows into another surface. Magnitude can be between 0 and 1 where 0 is flat and 1 curves the most.

Set Surface Properties Before and After Creation

Set defaults that control a variety of surface properties before and after you create the surface objects.

- Surface Modeling System Variables. There are a number of system variables that are frequently used and changed during surface creation: SURFACEMODELINGMODE, SURFACEASSOCIATIVITY, SURFACEASSOCIATIVITYDRAG, SURFACEAUTOTRIM, and SUBOBJSELECTIONMODE.

- Properties palette. Modifies properties for both the surface objects and their subobjects after they are created. For example, you can change the number of isolines in the U and V directions.

Quick Reference

Commands

3DOSNAP

Sets the object snap modes for 3D objects.

ANALYSISZEBRA

Projects stripes onto a 3D model to analyze surface continuity.

BREP

Removes the history from 3D solids and composite solids, and associativity from surfaces.

CONVTONURBS

Converts 3D solids and surfaces into NURBS surfaces.

CONVTOSURFACE

Converts objects to 3D surfaces.

EXTRUDE

Creates a 3D solid or surface by extending the dimensions of an object.

JOIN

Joins the endpoints of linear and curved objects to create a single object.

LOFT

Creates a 3D solid or surface in the space between several cross sections.

PLANESURF

Creates a planar surface.

REVOLVE

Creates a 3D solid or surface by sweeping an object around an axis.

SURFBLEND

Creates a continuous blend surface between two existing surfaces.

SURFNETWORK

Creates a surface in the space between several curves in the U and V directions (including surface and solid edge subobjects).

SURFOFFSET

Creates a parallel surface a specified distance from the original surface.

SURFPATCH

Creates a new surface by fitting a cap over a surface edge that forms a closed loop.

SWEEP

Creates a 3D solid or surface by sweeping a 2D or 3D object or subobject along a path.

VISUALSTYLES

Creates and modifies visual styles and applies a visual style to a viewport.

System Variables

3DOSMODE

Controls the settings for the 3D object snaps.

DELOBJ

Controls whether geometry used to create other objects is retained or deleted.

FACETRES

Adjusts the smoothness of shaded and rendered objects and objects with hidden lines removed.

ISOLINES

Specifies the number of contour lines displayed on the curved surfaces of 3D solids.

SUBOBJSELECTIONMODE

Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

SURFACEASSOCIATIVITY

Controls whether surfaces maintain a relationship with the objects from which they were created.

SURFACEASSOCIATIVITYDRAG

Sets the dragging preview behavior of associative surfaces.

SURFACEMODELINGMODE

Controls whether surfaces are created as procedural surfaces or NURBS surfaces.

Create Procedural Surfaces

Create procedural surfaces by blending, patching, and offsetting existing surfaces or by converting 3D solids, meshes and other planar geometry.

Create Surfaces from Other Surfaces

There are many ways to create procedural surfaces from existing surfaces. These include blending, patching, and offsetting or creating network and planar surfaces.

When you create procedural surfaces, use SURFACEASSOCIATIVITY to work with the surfaces as a group. Just as you can modify the face of a solid and have the entire solid update, when you modify a group of surfaces that are associative, all of the surfaces update accordingly.

Create Planar Surfaces

Create planar surfaces in the space between edge subobjects, splines and other 2D and 3D curves.

Create planar surfaces with the PLANESURF command Planar surfaces can be created from multiple closed objects and the curves can be surface or solid edge subobjects. During creation, specify the tangency and bulge magnitude.

See also:

- Overview of Creating Surfaces on page 1080

To create a planar surface

1. Click Surface tab ➤ Create panel ➤ Planar. 

2. Click and drag an area onscreen.
3. Hold Shift and press the mouse wheel to orbit and inspect the planar surface.

Quick Reference

Commands

3DOSNAP

Sets the object snap modes for 3D objects.

LIST

Displays property data for selected objects.

MEASUREGEOM

Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

PLANESURF

Creates a planar surface.

UCS

Sets the origin and orientation of the current UCS.

VISUALSTYLES

Creates and modifies visual styles and applies a visual style to a viewport.

System Variables

3DOSMODE

Controls the settings for the 3D object snaps.

DELOBJ

Controls whether geometry used to create other objects is retained or deleted.

LIST

Displays property data for selected objects.

MEASUREGEOM

Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

SUBOBJSELECTIONMODE

Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

SURFACEASSOCIATIVITY

Controls whether surfaces maintain a relationship with the objects from which they were created.

Create Network Surfaces

Create non-planar surfaces in the space between edge subobjects, splines and other 2D and 3D curves.

Create non-planar, network surfaces with the SURFNETWORK command. Network surfaces are similar to lofted surfaces in that they are created in the space between several curves in the U and V directions. The curves can be surface or solid edge subobjects. When you create the surface you can specify the tangency and bulge magnitude of the surface edges.

See also:

- Overview of Creating Surfaces on page 1080

To create a network surface



1. Click Surface tab ➤ Create panel ➤ Network.
2. In the drawing area, select cross section curves in the first direction (U or V) and press Enter.
3. Select cross sections in the second direction and press Enter.

Quick Reference

Commands

3DOSNAP

Sets the object snap modes for 3D objects.

LIST

Displays property data for selected objects.

MEASUREGEOM

Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

SURFNETWORK

Creates a surface in the space between several curves in the U and V directions (including surface and solid edge subobjects).

UCS

Sets the origin and orientation of the current UCS.

VISUALSTYLES

Creates and modifies visual styles and applies a visual style to a viewport.

System Variables

3DOSMODE

Controls the settings for the 3D object snaps.

DELOBJ

Controls whether geometry used to create other objects is retained or deleted.

LIST

Displays property data for selected objects.

MEASUREGEOM

Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

SUBOBJSELECTIONMODE

Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

SURFACEASSOCIATIVITY

Controls whether surfaces maintain a relationship with the objects from which they were created.

Blend a Surface

Create a transition surface between two existing surfaces.

Use SURFBLEND to create a new surface between existing surfaces and solids. When you blend surfaces together, specify the surface continuity and bulge magnitude for the start and end edges.

See also:

- Overview of Creating Surfaces on page 1080

To create a blend surface between a surface and a solid

1. Click Surface tab ➤ Create panel ➤ Blend. 
2. Click Home tab ➤ Selection panel ➤ Edge. 

   NOTE If the drawing area is very busy, use the CULLINGOBJ and the CULLINGOBJSELECTION to further filter out unneeded objects.

3. In the drawing area, select a surface edge subobject and a solid edge subobject.
4. Specify the continuity and bulge magnitude for each edge and press Enter.

Quick Reference

Commands

3DOSNAP

Sets the object snap modes for 3D objects.

LIST

Displays property data for selected objects.

MEASUREGEOM

Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

SURFBLEND

Creates a continuous blend surface between two existing surfaces.

UCS

Sets the origin and orientation of the current UCS.

VISUALSTYLES

Creates and modifies visual styles and applies a visual style to a viewport.

System Variables

3DOSMODE

Controls the settings for the 3D object snaps.

CULLINGOBJ

Controls whether 3D subobjects that are hidden from view can be highlighted or selected.

CULLINGOBJSELECTION

Controls whether 3D objects that are hidden from view can be highlighted or selected.

DELOBJ

Controls whether geometry used to create other objects is retained or deleted.

PREVIEWCREATIONTRANSPARENCY

Controls the transparency of the preview generated while using SURFBLEND, SURFPATCH, SURFFILLET, FILLETEDGE, CHAMFEREDGE, and LOFT.

SUBOBJSELECTIONMODE

Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

SURFACEASSOCIATIVITY

Controls whether surfaces maintain a relationship with the objects from which they were created.

Patch a Surface

Create a surface by patching a closed surface or curve.

Use SURFPATCH to create a surface inside a closed curve (such as a closed spline) that is another surface's edge. You can also draw a guide curve to constrain the shape of the patch surface with the constrain geometry option. When you patch surfaces, specify the continuity and bulge magnitude.

See also:

- Overview of Creating Surfaces on page 1080

To patch the top of a closed surface

1. Click Surface tab ➤ Create panel ➤ Patch. 
2. Click Home tab ➤ Selection panel ➤ Edge. 

   NOTE If the drawing area is very busy, use the CULLINGOBJ and the CULLINGOBJSELECTION to further filter out unneeded objects.

3. In the drawing area, select a surface edge subobject.
4. Specify the continuity and bulge magnitude and press Enter.

Quick Reference

Commands

3DOSNAP

Sets the object snap modes for 3D objects.

LIST

Displays property data for selected objects.

MEASUREGEOM

Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

SURFPATCH

Creates a new surface by fitting a cap over a surface edge that forms a closed loop.

UCS

Sets the origin and orientation of the current UCS.

VISUALSTYLES

Creates and modifies visual styles and applies a visual style to a viewport.

System Variables

3DOSMODE

Controls the settings for the 3D object snaps.

CULLINGOBJ

Controls whether 3D subobjects that are hidden from view can be highlighted or selected.

CULLINGOBJSELECTION

Controls whether 3D objects that are hidden from view can be highlighted or selected.

DELOBJ

Controls whether geometry used to create other objects is retained or deleted.

PREVIEWCREATIONTRANSPARENCY

Controls the transparency of the preview generated while using SURFBLEND, SURFPATCH, SURFFILLET, FILLETEDGE, CHAMFEREDGE, and LOFT.

SUBOBJSELECTIONMODE

Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

SURFACEASSOCIATIVITY

Controls whether surfaces maintain a relationship with the objects from which they were created.

Offset a Surface

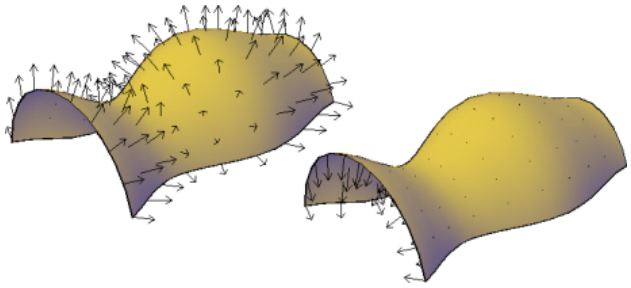
Create a parallel surface a specified distance from the original surface.

With SURFOFFSET specify the offset distance and whether or not the offset surface maintains associativity with the original surface. You can also specify the offset distance with a mathematical expression. See Constrain a Design with Formulas and Equations on page 903.

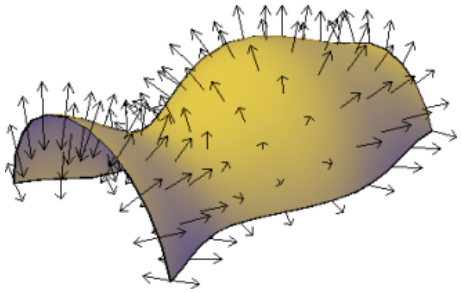
Surface Offset Options

When you offset a surface, you can do the following:

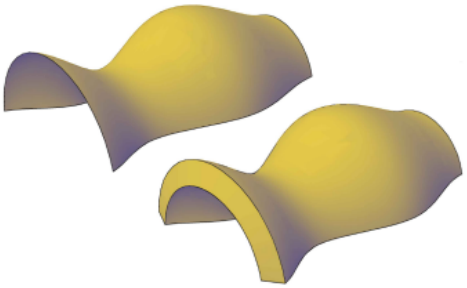
- Change the offset direction with the Flip option



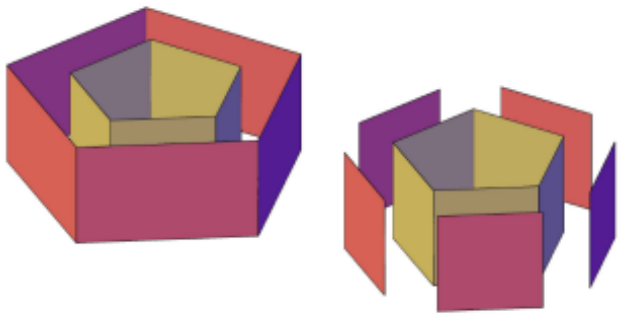
- Offset in both directions to create two new surfaces



- Create a solid between the offset surfaces



- If you are offsetting more than one surface, you can specify whether the offset surfaces remain connected.



- Enter an expression that will constrain the distance of the offset surface to the original surface. This option only appears if associativity is on. See Constrain a Design with Formulas and Equations on page 903.

See also:

- Overview of Creating Surfaces on page 1080

To create a solid from an offset surface

1. Click Surface tab ➤ Create panel ➤ Offset. 
2. In the drawing area, select a surface and press Enter. Arrows display to show the offset direction.
3. Enter **s** and press Enter.
4. Enter an offset distance and press Enter.

The surface creates a parallel surface and connects the two to form a 3D solid object.

Quick Reference

Commands

3DOSNAP

Sets the object snap modes for 3D objects.

LIST

Displays property data for selected objects.

MEASUREGEOM

Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

SURFOFFSET

Creates a parallel surface a specified distance from the original surface.

UCS

Sets the origin and orientation of the current UCS.

VISUALSTYLES

Creates and modifies visual styles and applies a visual style to a viewport.

System Variables

3DOSMODE

Controls the settings for the 3D object snaps.

CULLINGOBJ

Controls whether 3D subobjects that are hidden from view can be highlighted or selected.

CULLINGOBJSELECTION

Controls whether 3D objects that are hidden from view can be highlighted or selected.

DELOBJ

Controls whether geometry used to create other objects is retained or deleted.

SUBOBJSELECTIONMODE

Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

SURFACEASSOCIATIVITY

Controls whether surfaces maintain a relationship with the objects from which they were created.

Convert Objects to Procedural Surfaces

Convert 3D solids, meshes, and 2D geometry to procedural surfaces.

Use CONVTOSURFACE to convert any of the following objects into surfaces:

- 2D solids
- Meshes
- Regions
- Open, zero-width polylines with thickness
- Lines with thickness
- Arcs with thickness
- Planar 3D faces

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.

Quick Reference

Commands

CONVTOSURFACE

Converts objects to 3D surfaces.

Create NURBS Surfaces

See also:

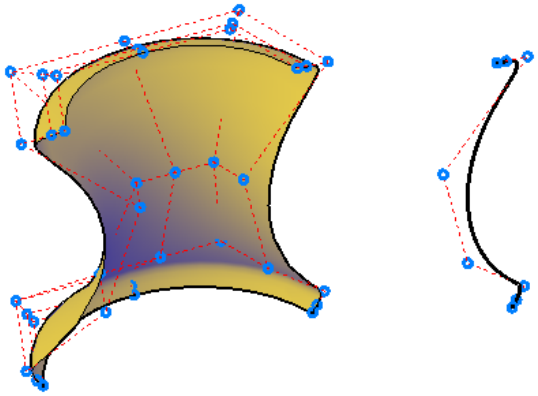
- Create Solids and Surfaces from Lines and Curves on page 1030
- Create Surfaces from Other Surfaces on page 1086
- Edit NURBS Surfaces on page 1225
- Rebuild NURBS Surfaces and Curves on page 1227

- Draw Splines on page 723
- Modify Splines on page 853

Create NURBS surfaces by enabling NURBS creation and using many of the same commands used to create procedural surfaces. You can also convert existing procedural surfaces into NURBS surfaces.

NURBS (Non-Uniform Rational B-Splines) surfaces are part of the suite of 3D modeling objects that AutoCAD offers (in addition to 3D solids, procedural surfaces, and meshes).

NURBS surfaces are based on Bezier curves or splines. Therefore, settings such as degree, fit points, control vertices, weights, and knot parameterization are important in defining a NURBS surface or curve. AutoCAD splines are optimized to create NURBS surfaces allowing you to control many of these options (see SPLINE and SPLINEDIT). The illustration below shows the control vertices that display when you select a NURBS surface or spline.



Two Methods for Creating NURBS Surfaces

There are two ways to create NURBS surfaces:

- **SURFACEMODELINGMODE system variable** - Use any of the surface creation commands while this system variable is set to 1.
- **CONVTONURBS command** - Convert any existing surfaces with this command.

It is important to plan ahead with NURBS modeling, since NURBS surfaces cannot be converted back into procedural surfaces.

To create a NURBS surface by lofting

1 Click Surface tab ➤ Create panel ➤ NURBS Creation  (if it is not already enabled).

2 Click Surface tab ➤ Create panel ➤ Loft. 

3 Select the cross section profiles and press *Enter*. Select them in the order in which you want the new 3D object to pass through them.

After the loft operation, the original objects are deleted or retained, depending on the setting of the DELOBJ system variable.

To convert a solid into a NURBS surface

1 Click Surface tab ➤ Control Vertices panel ➤ Convert to NURBS. 

2 Select a solid and press *Enter*. The object is converted into multiple NURBS surfaces (each face becomes a separate surface).

NOTE To display the control vertices, click Surface tab ➤ Control Vertices panel ➤ Show CV. 

To convert a mesh object into a NURBS surface

1 Click Mesh tab ➤ Convert Mesh ➤ Convert to Surface. 

2 Select a mesh object and press *Enter*.

The object is converted to a procedural surface.

3 Click Surface tab ➤ Create panel ➤ NURBS Creation. 

The object is converted to a NURBS surface.

NOTE To display the control vertices, click Surface tab ➤ Control Vertices panel ➤ Show CV.

Quick Reference

Commands

CONVTONURBS

Converts 3D solids and surfaces into NURBS surfaces.

CVSHOW

Displays the control vertices for specified NURBS surfaces or curves.

EXTRUDE

Creates a 3D solid or surface by extending the dimensions of an object.

LIST

Displays property data for selected objects.

LOFT

Creates a 3D solid or surface in the space between several cross sections.

MEASUREGEOM

Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

REVOLVE

Creates a 3D solid or surface by sweeping an object around an axis.

SPLINE

Creates a smooth curve that passes through or near a set of fit points, or that is defined by the vertices in a control frame.

SPLINEDIT

Modifies the parameters of a spline or converts a spline-fit polyline to a spline.

SURFBLEND

Creates a continuous blend surface between two existing surfaces.

SURFEXTEND

Lengthens a surface by a specified distance.

SURFFILLET

Creates a filleted surface between two other surfaces.

SURFNETWORK

Creates a surface in the space between several curves in the U and V directions (including surface and solid edge subobjects).

SURFOFFSET

Creates a parallel surface a specified distance from the original surface.

SURFPATCH

Creates a new surface by fitting a cap over a surface edge that forms a closed loop.

SWEEP

Creates a 3D solid or surface by sweeping a 2D or 3D object or subobject along a path.

System Variables

SURFACEMODELINGMODE

Controls whether surfaces are created as procedural surfaces or NURBS surfaces.

Using the Spline tool to create NURBS Surfaces and Splines

See also:

- Draw Splines on page 723
- Modify Splines on page 853

The spline tool is optimized to work with NURBS modeling

NURBS surfaces can be created from a number of 2D objects, including edge subobjects, polylines, and arcs. But the spline tool is the only object that has options that are compatible with creating NURBS surface. Not only do splines consists of Bezier arcs, but they also can be defined with both control vertices

and fit points. The fit points and control vertices offer different editing options such as knot parameterization and degree options.

Moving Fit Points vs. Moving Control Vertices

NURBS curves have both fit points and control vertices. The fit points lie on the line, and the control vertices lay outside the line. Use fit points to make a change to one small part of a curve; use control vertices to make changes that will affect the shape of the curve as a whole.

Clamp Surfaces and Curves with Open and Closed Geometry

NURBS surfaces and curves can have a clamp, closed, or open form. The form affects how the object deforms.

- **Open Curves and surfaces** - have their start and end CVs in different positions - it doesn't form a loop. If you snap the start and end CVs of an open curve to the same position, it's still an open curve, because you can still drag these points away from one another.
- **Closed Curves and Surfaces** - a loop with coinciding start and end CVs. Where they meet is called a seam. If you move one CV, the other moves with it.
- **Clamp Curve** - is a closed loop with a seam that creates extra, unseen CVs. These unseen CVs can cause the shape to wrinkle and crease when it is reshaped.

Quick Reference

Commands

SPLINE

Creates a smooth curve that passes through or near a set of fit points, or that is defined by the vertices in a control frame.

SPLINEDIT

Modifies the parameters of a spline or converts a spline-fit polyline to a spline.

System Variables

SURFACEMODELINGMODE

Controls whether surfaces are created as procedural surfaces or NURBS surfaces.

Create Associative Surfaces

Associative surfaces automatically adjust to changes made to other, related objects.

When surface associativity is on, surfaces are created with a relationship to the surface or profiles that created them.

Associativity allows you to:

- Reshape the generating profiles to automatically reshape the surface.
- Work with a group of surfaces as if they were one object. Just as reshaping one face of a solid box adjusts the entire primitive, reshaping one surface or edge in a group of associated surfaces adjusts the entire group.
- Use geometric constraints on the 2D profiles of a surface.
- Assign mathematical expressions to derive properties of surfaces, such as height and radius. For example, specify that the height of an extruded surface be equal to one half the lengh of another object.

As you add more objects and edit them, all these objects become related and create a chain of dependency. Editing one object can ripple through and affect all associated objects.

It is important to understand the chain of associativity because moving or deleting one of the links in the chain can break the relationship between all the objects.

NOTE To modify the shape of a surface that is generated from a curve or spline, you must select and modify the generating curve or spline, not the surface itself. If you modify the surface itself, you will lose associativity.

When associativity is on, the DELOBJ system variable is ignored. If **Surface Associativity** and NURBS Creation are both on, surface are created as NURBS surfaces, not associative surfaces.

Save time by planning your model ahead; you cannot go back and add associativity after the model has been created. Also, be careful not to accidentally break associativity by dragging objects away from the group.

To create an associative surface

- Click Surface tab ➤ Create panel ➤ Surface Associativity.
  Any new procedural surfaces will be associative.

> **NOTE** NURBS creation overrides Surface Associativity. If Surface Associativity and NURBS Creation are both on, surface associativity will not work.

To see what objects a surface is associated with

1. In a drawing, select an associative surface.
2. Open the properties palette, in Surface Associativity, select **Show** from the Show Associativity drop-down list.
3. Roll over the surface and nearby objects.

The associated objects, such as generating curves or edge subobjects highlight as well as the surface itself.

To turn associativity off for a particular surface

1. In a drawing, select an associative surface.
2. Open the properties palette, in Surface Associativity, select **Non** from the Maintain Associativity drop-down list.

The surface maintains its associativity to other objects. But any new objects that are created will not be associated with this surface. It breaks the chain of associativity.

To remove associativity from a surface

1. In a drawing, select an associative surface.
2. Open the properties palette, in Surface Associativity, select **Remove** from the Maintain Associativity drop-down list.

The surface becomes a generic surface. You can no longer change any or its properties in the properties palette and it loses its relationship with other objects.

Quick Reference

Commands

BREP

Removes the history from 3D solids and composite solids, and associativity from surfaces.

EXTRUDE

Creates a 3D solid or surface by extending the dimensions of an object.

JOIN

Joins the endpoints of linear and curved objects to create a single object.

LIST

Displays property data for selected objects.

LOFT

Creates a 3D solid or surface in the space between several cross sections.

MEASUREGEOM

Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

PLANESURF

Creates a planar surface.

PROJECTGEOMETRY

Projects points, lines, or curves onto a 3D solid or surface from different directions.

REVOLVE

Creates a 3D solid or surface by sweeping an object around an axis.

SURFBLEND

Creates a continuous blend surface between two existing surfaces.

SURFNETWORK

Creates a surface in the space between several curves in the U and V directions (including surface and solid edge subobjects).

SURFOFFSET

Creates a parallel surface a specified distance from the original surface.

SURFPATCH

Creates a new surface by fitting a cap over a surface edge that forms a closed loop.

SWEEP

Creates a 3D solid or surface by sweeping a 2D or 3D object or subobject along a path.

VISUALSTYLES

Creates and modifies visual styles and applies a visual style to a viewport.

System Variables

SURFACEASSOCIATIVITY

Controls whether surfaces maintain a relationship with the objects from which they were created.

SURFACEASSOCIATIVITYDRAG

Sets the dragging preview behavior of associative surfaces.

Create Geometric Relationships between Associative Surfaces

Use geometric constraints to constrain and restrict surfaces. And use mathematical expressions to derive surface properties.

Use Geometric Constraints with Surface Profiles

Just as with 2D drafting, geometric constraints can be used to restrict the movement of 3D surfaces. For example, you can specify that a surface remain fixed in a perpendicular or parallel location to another object. In the example below, an offset surface is locked in a parallel position to its original surface.

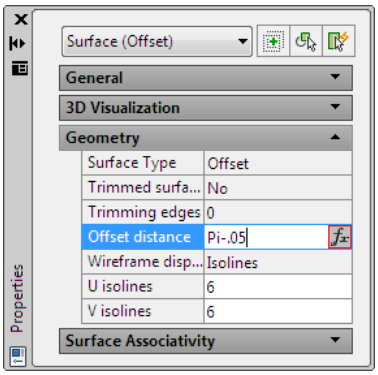


Constraints are applied to the 2D profile object used to create the surface, not the surface itself. Use selection cycling to be sure that you are selecting the

profile curve and not the surface or the edge subobject. See Apply or Remove Geometric Constraints on page 872.

Use Mathematical Expressions to Derive Surface Properties

Dimensional constraints are user-defined expressions that are applied in the Properties palette for that surface.



For a complete list of operators and functions allowed in expressions, see Control Geometry with the Parameters Manager on page 905. The following table lists the surface types and their properties that accept expressions

Type of Surface	Surface properties that can be constrained.
Blend Surface	Bulge Magnitude
Extend Surface	Extension Distance
Extruded Surface	■ Height ■ Taper
Fillet Surface	Fillet Radius
Loft Surface	Bulge Magnitude
Network Surface	Bulge Magnitude
Offset Surface	Offset Distance
Patch Surface	Bulge Magnitude

Type of Surface	Surface properties that can be constrained.
Revolve Surface	Revolve Angle

To create a parallel relationship between two surfaces

1. Enter **dsettings** and select the Selection Cycling tab.
2. Click Allow Selection Cycling to turn selection cycling on.
3. Enter **geomconstraint**. Enter **pa** to select the parallel option.
4. Select the first and second profile objects. Be careful not to select the surface or the edge subobject.

If the surfaces were not already parallel, they are moved into a parallel location and the parallel constraint symbol appears.

To add a user expression to specify the height of a surface

1. Right-click on a surface and select **Properties.**

   The properties palette displays.
2. In the **Geometry** section, enter an expression in the Height field.

   A symbol appears next to that field to indicate that a dimensional constraint is applied to this property.

Quick Reference

Commands

AUTOCONSTRAIN

Applies geometric constraints to a selection set of objects based on orientation of the objects relative to one another.

CONSTRAINTBAR

Displays or hides the geometric constraints on an object.

CONSTRAINTSETTINGS

Controls the display of geometric constraints on constraint bars.

DCDISPLAY

Displays or hides the dynamic constraints associated with a selection set of objects.

DELCONSTRAINT

Removes all geometric and dimensional constraints from a selection set of objects.

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects, or converts associative dimensions to dimensional constraints.

GEOMCONSTRAINT

Displays or hides the geometric constraints on an object.

LIST

Displays property data for selected objects.

MEASUREGEOM

Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

PARAMETERS

Opens the Parameters Manager palette that includes all dimensional constraint parameters, reference parameters, and user variables in the current drawing.

-PARAMETERS

Opens the Parameters Manager palette that includes all dimensional constraint parameters, reference parameters, and user variables in the current drawing.

PARAMETERSCLOSE

Closes the Parameters Manager palette.

TEXTEDIT

Edits a selected multiline or single-line text object, or the text in a dimension object.

System Variables

CCONSTRAINTFORM

Controls whether annotational or dynamic constraints are applied to objects.

CONSTRAINTBARDISPLAY

Controls the display of constraint bars after you apply constraints and when you select geometrically constrained drawings.

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.

DYNCONSTRAINTMODE

Displays hidden dimensional constraints when constrained objects are selected.

PARAMETERCOPYMODE

Controls how constraints and referenced user parameters are handled when constrained objects are copied between drawings, Model space and layouts, and block definitions.

PARAMETERSSTATUS

Indicates whether the Parameters Manager is displayed or hidden.

SURFACEASSOCIATIVITY

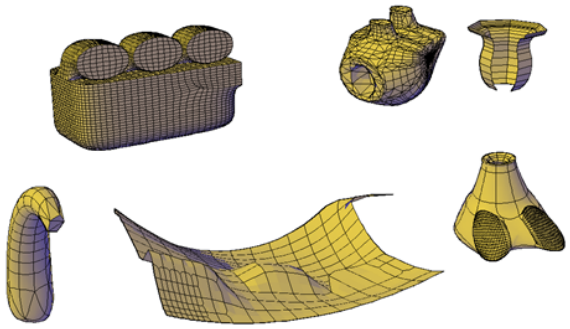
Controls whether surfaces maintain a relationship with the objects from which they were created.

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 , 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

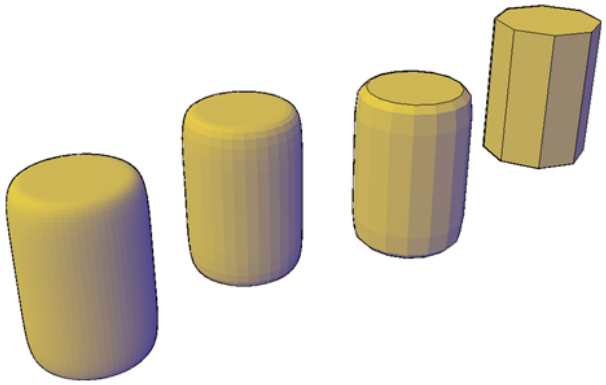
you specify. Although you can continue to create legacy polygonal and polyface meshes, it is recommended that you convert to the enhanced mesh object type to obtain enhanced editing capabilities.

About Tessellation

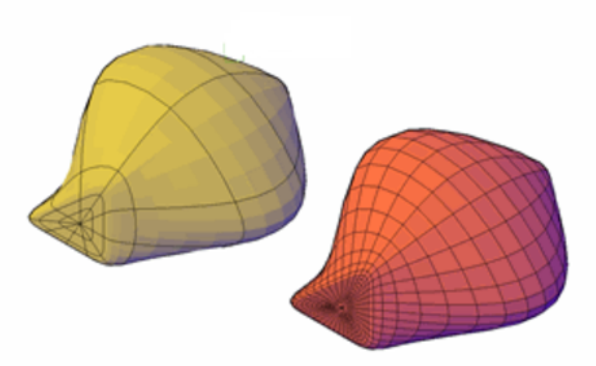
Tessellation is a collection of planar shapes that tile a mesh object. The tessellation divisions, visible in unselected mesh objects, mark the edges of the editable mesh faces. (To see these divisions in the Hidden or Conceptual visual styles, VSEDGES must be set to 1.)

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 1238

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.

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

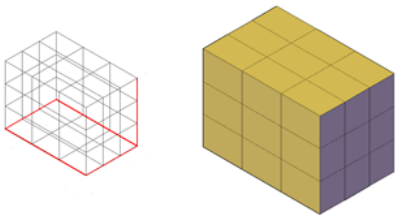
Controls the types of edges that are displayed in the viewport.

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 (work plane).

The following system variables are used to control the number of divisions for each dimension of a new mesh box:

- DIVMESHBOXHEIGHT
- DIVMESHBOXLENGTH
- DIVMESHBOXWIDTH

You can also use the Mesh Primitive Options dialog box to set the default number of divisions for a mesh box.

After a mesh primitive is created, the current level of smoothness for the object can be modified.

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.

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.

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.

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.

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

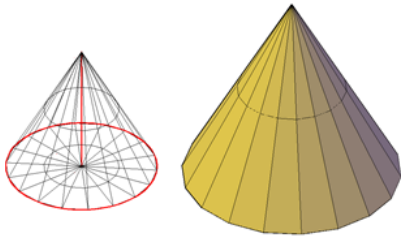
Sets the number of subdivisions for the width of a mesh box along the *Y* axis.

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.

The following system variables are used to control the number of divisions for each dimension of a new mesh cone:

- DIVMESHCONEAXIS
- DIVMESHCONEBASE
- DIVMESHCONEHEIGHT

You can also use the Mesh Primitive Options dialog box to set the default number of divisions for a mesh cone.

After a mesh primitive is created, the current level of smoothness for the object can be modified.

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.

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.

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.

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.

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

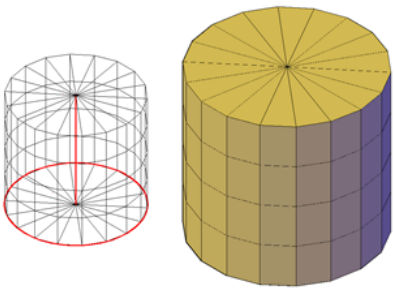
Sets the number of subdivisions between the base and the point or top of the mesh cone.

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.

The following system variables are used to control the number of divisions for each dimension of a new mesh cylinder:

- DIVMESHCYLAXIS

- DIVMESHCYLBASE
- DIVMESHCYLHEIGHT

You can also use the Mesh Primitive Options dialog box to set the default number of divisions for a mesh cylinder.

After a mesh primitive is created, the current level of smoothness for the object can be modified.

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.

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.

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.

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

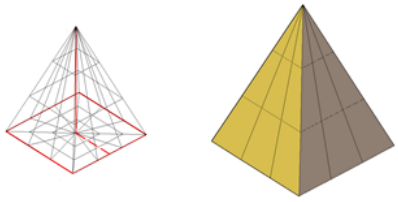
Sets the number of subdivisions between the base and the top of the mesh cylinder.

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.

The following system variables are used to control the number of divisions for each dimension of a new mesh pyramid:

- DIVMESHPYRBASE
- DIVMESHPYRHEIGHT
- DIVMESHPYRLENGTH

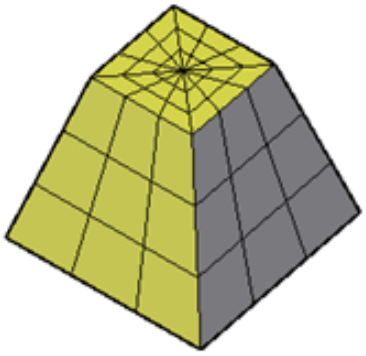
You can also use the Mesh Primitive Options dialog box to set the default number of divisions for a mesh pyramid.

After a mesh primitive is created, the current level of smoothness for the object can be modified.

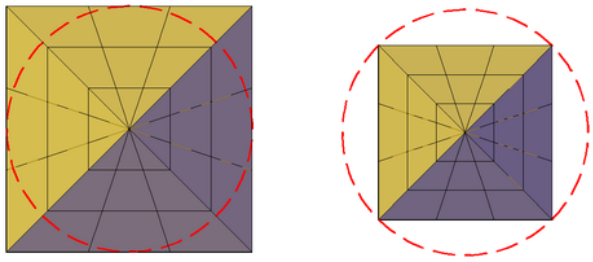
Mesh Pyramid Creation Options

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.

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.

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

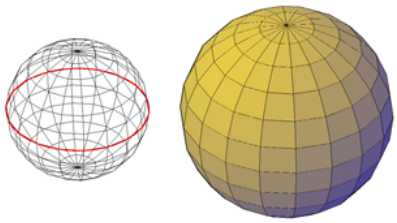
Sets the number of subdivisions along each dimension of a mesh pyramid base.

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).

The following system variables are used to control the number of divisions for each dimension of a new mesh sphere:

- DIVMESHSPHEREAXIS
- DIVMESHSPHEREHEIGHT

You can also use the Mesh Primitive Options dialog box to set the default number of divisions for a mesh sphere.

After a mesh primitive is created, the current level of smoothness for the object can be modified.

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.

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.

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

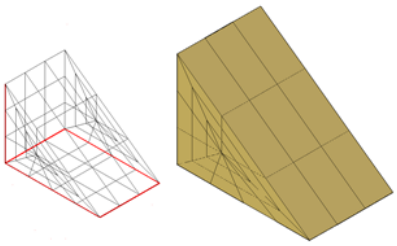
Sets the number of subdivisions between the two axis endpoints of the mesh sphere.

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.

The following system variables are used to control the number of divisions for each dimension of a new mesh wedge:

- DIVMESHWEDGEBASE
- DIVMESHWEDGEHEIGHT
- DIVMESHWEDGELENGTH
- DIVMESHWEDGESLOPE
- DIVMESHWEDGEWIDTH

You can also use the Mesh Primitive Options dialog box to set the default number of divisions for a mesh wedge.

After a mesh primitive is created, the current level of smoothness for the object can be modified.

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.

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.

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.

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.

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

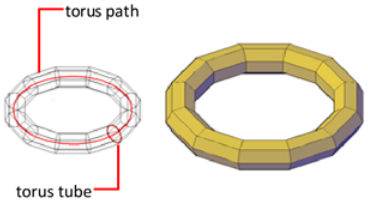
Sets the number of subdivisions for the width of the mesh wedge along the Y axis.

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.

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.

The following system variables are used to control the number of divisions for each dimension of a new mesh torus:

- DIVMESHTORUSPATH
- DIVMESHTORUSSECTION

You can also use the Mesh Primitive Options dialog box to set the default number of divisions for a mesh torus.

After a mesh primitive is created, the current level of smoothness for the object can be modified.

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.

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.

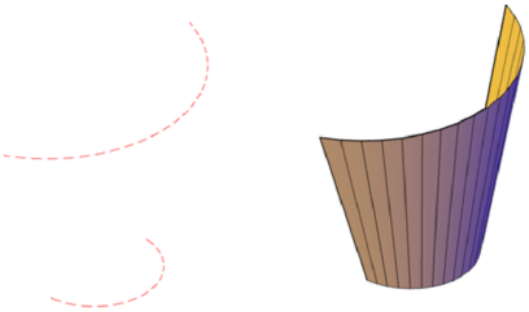
You can use a variety of methods to create mesh objects whose edges are defined by other objects. The MESHTYPE system variable controls whether the new objects are valid mesh objects, or whether they are created using legacy polyface or polygon geometry.

You can control whether the mesh is displayed as a wireframe, hidden, or conceptual image by changing the visual style (VISUALSTYLES).

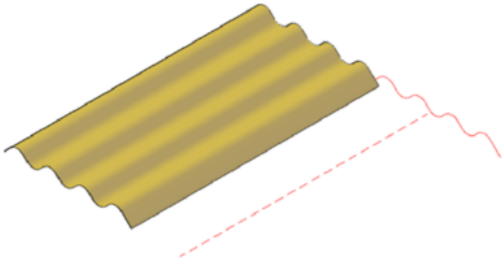
Types of Meshes Created from Other Objects

You can create several types of meshes that are based on existing objects.

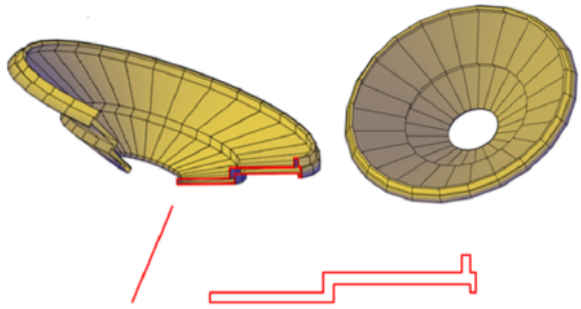
- **Ruled mesh**. RULESURF creates a mesh that represents the ruled surface between two lines or curves.



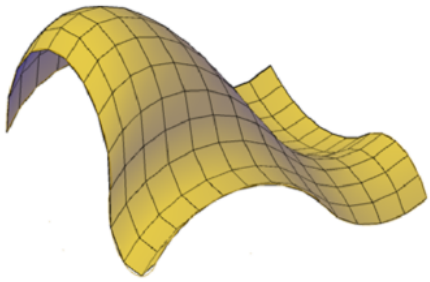
- **Tabulated mesh**. TABSURF creates a mesh that represents a general tabulated surface. The surface is defined by the extrusion of a line or curve (called a path curve) in a specified direction and distance (called a direction vector or path).



- **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).

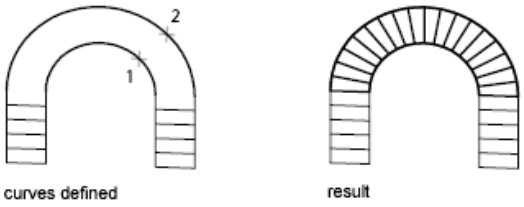


Create a Ruled Mesh

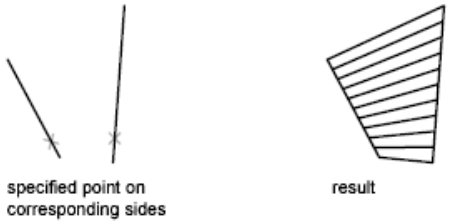
There are several methods for creating meshes.

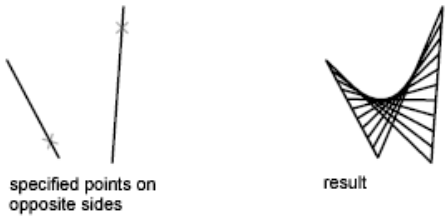
With RULESURF, you create a mesh between two lines or curves. Use two different objects to define the edges of the ruled mesh: lines, points, arcs, circles, ellipses, elliptical arcs, 2D polylines, 3D polylines, or splines.

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.

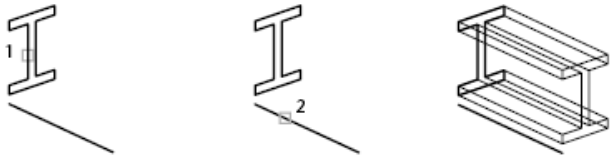




Create a Tabulated Mesh

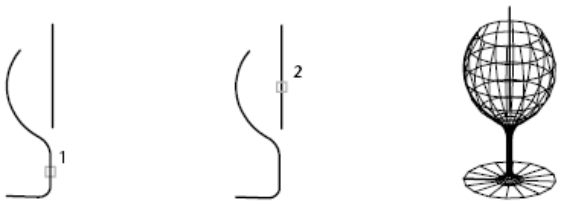
With the TABSURF command, you can create a mesh that represents a general tabulated surface defined by a path curve and a direction vector. The path curve can be a line, arc, circle, ellipse, elliptical arc, 2D polyline, 3D polyline, or spline. The direction vector can be a line or an open 2D or 3D polyline.

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.

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.

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.

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.

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.

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.

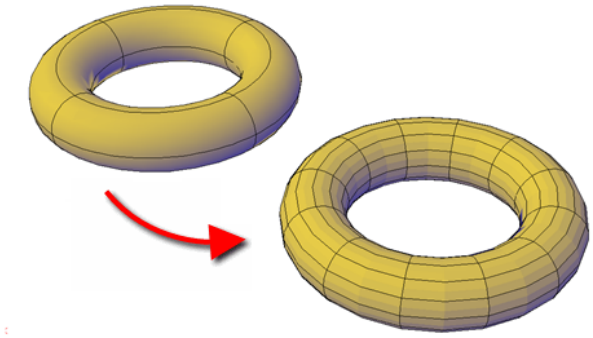
SURFV

Sets the surface density for PEDIT Smooth in the N direction and the V isolines density on surface objects.

Create Meshes 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 following system variables:

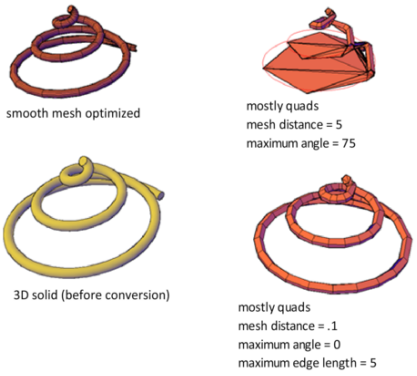
- FACETERDEVNORMAL
- FACETERDEVSURFACE
- FACETERGRIDRATIO
- FACETERMAXEDGELENGTH
- FACETERMAXGRID

- FACETERMESHTYPE
- FACETERMINUGRID
- FACETERMINVGRID
- FACETERPRIMITIVEMODE
- FACETERSMOOTHLEV

Settings can also be changed using the Mesh Tessellation Options dialog box.

For example, if the smooth mesh optimized mesh type (FACETERMESHTYPE system variable) 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 (FACETERMAXEDGELENGTH system varaible).

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.

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.

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.

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)

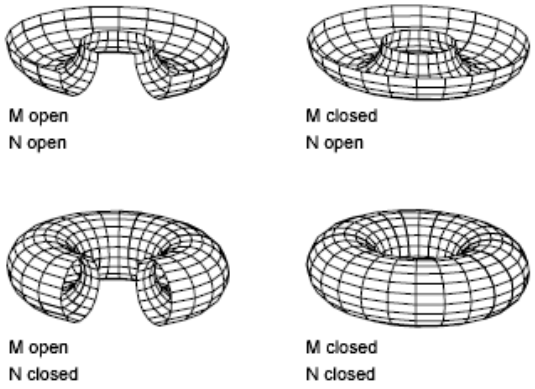
Create custom polygon or polyface mesh by specifying vertices.

Specify individual vertices when you create mesh using the 3DMESH, PFACE, and 3DFACE commands.

Understand Legacy Mesh Construction

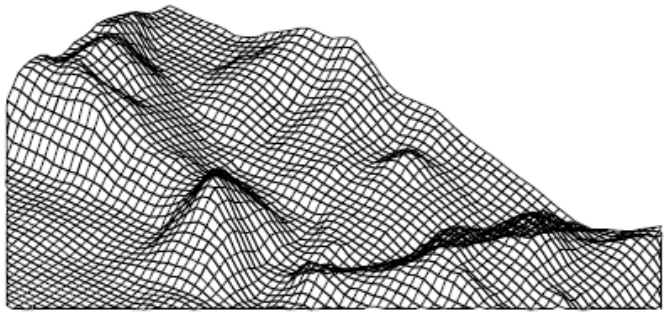
The mesh density controls the number of facets in legacy polygonal and polyface meshes. Density is defined in terms of a matrix of *M* and *N* vertices, like a grid consisting of columns and rows. *M* and *N* specify the column and row position, respectively, of any given vertex.

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.

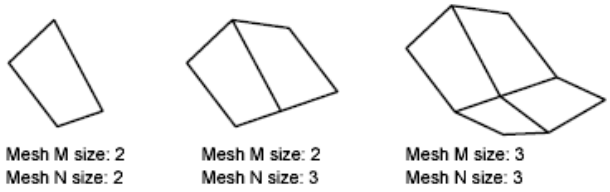


Create a Rectangular Mesh

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.

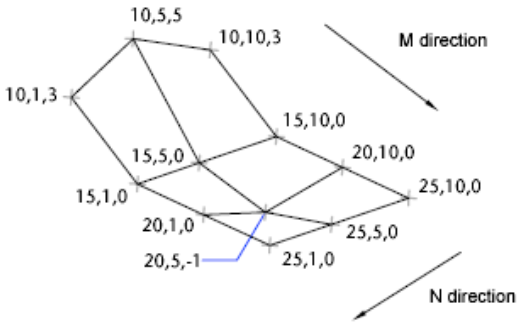


You can close the meshes with PEDIT. You can use 3DMESH to construct irregular meshes.

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.

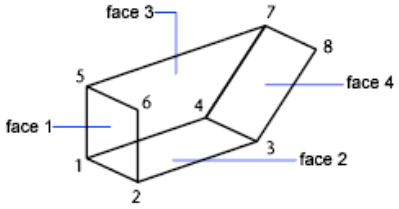
Creating a polyface mesh is like creating a rectangular mesh. To create a polyface mesh, you specify coordinates for its vertices. You then define each face by entering vertex numbers for all the vertices of that face. As you create

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.



Create Polyface Mesh Vertex by Vertex

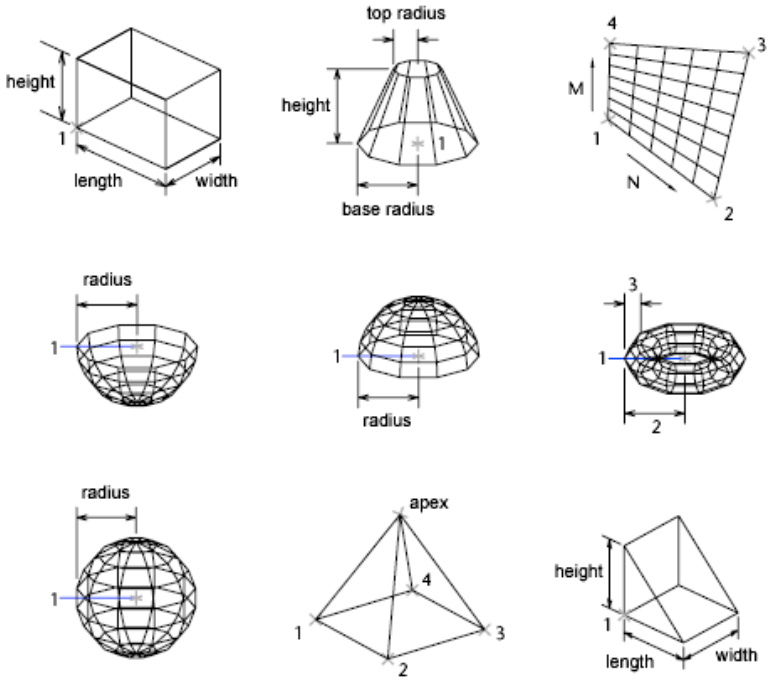
With the 3DFACE command, you can create three-dimensional polyface mesh by specifying each vertex. You can control visibility of each mesh edge segment.

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. At the Command prompt, enter **3dmesh**.
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.

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

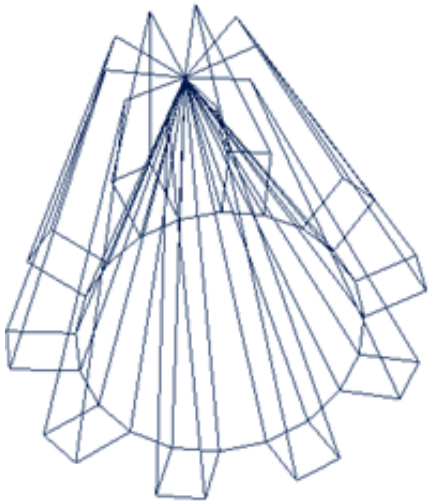
PFACEVMAX

Sets the maximum number of vertices per face.

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 work plane (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 work plane 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.

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.

Quick Reference

Commands

UCS

Sets the origin and orientation of the current UCS.

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.

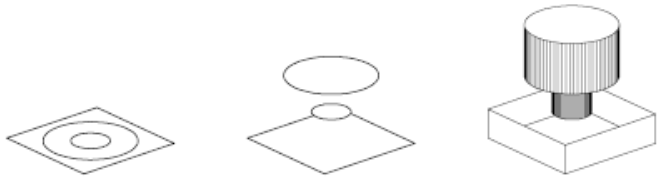
ISOLINES

Specifies the number of contour lines displayed on the curved surfaces of 3D solids.

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 Solids and Surfaces from Lines and Curves on page 1030

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.

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.

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

25

Change a 3D solid, surface or mesh with grips and gizmos, editing commands, properties, and by modifying its component subobjects.

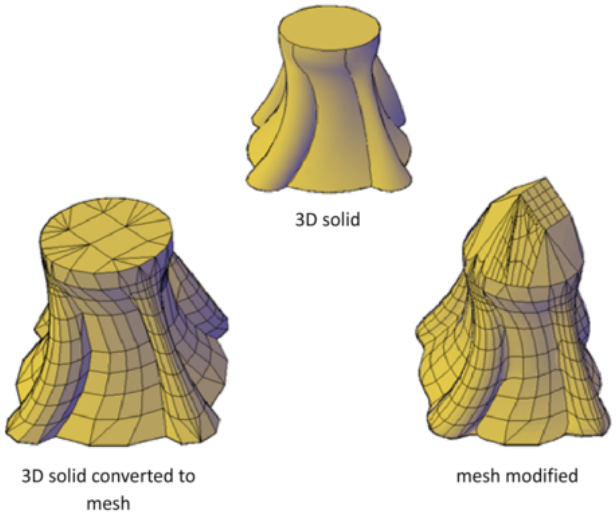
Overview of Modifying 3D Objects

3D 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, surfaces or meshes. Other methods are shared.

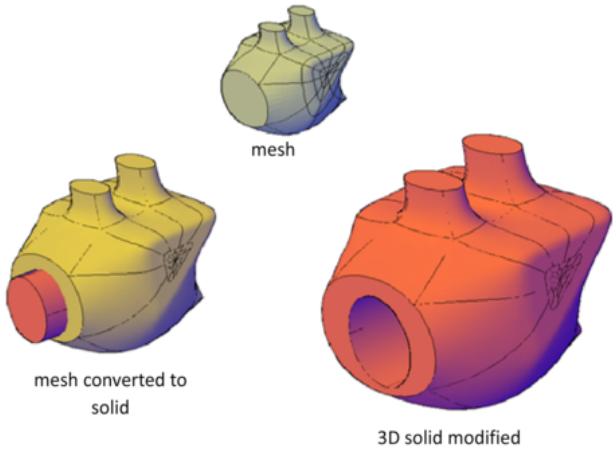
Convert to Other Object Types

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. For more information, see Overview of Coordinate Entry.
- **Rotate the view to display the model from different viewpoints.** Several navigation tools, including 3D Orbit and the ViewCube tool, are available to help you rotate around your workspace. For more information, see Use Viewing Tools on page 323.
- **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. For more information, see Display Multiple Views in Model Space on page 413.

See also:

- Control the User Coordinate System (UCS) on page 594
- Use Viewing Tools on page 323
- Display Multiple Views in Model Space on page 413

Use Gizmos to Modify Objects

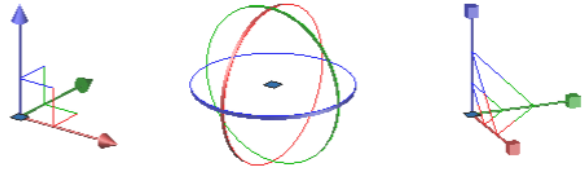
Use gizmos to move, rotate, or scale objects and subobjects in a 3D view.

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

Controls whether grips are automatically selected (made "hot") when subobjects are selected.

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

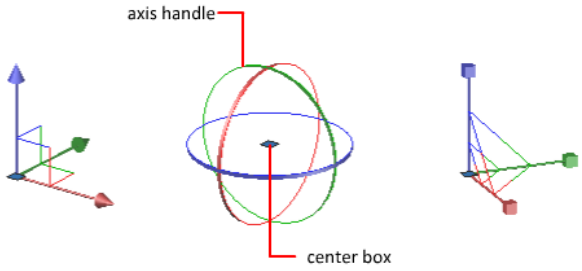
Gizmos help move, rotate, and scale 3D objects and subobjects.

Display the Gizmos

Gizmos are available only in 3D views that are set to use a 3D visual style such as 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 select a subobject on page 2321, 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.

To suppress the display of the 3D gizmo when an object is selected

- Click Home tab ➤ Subobject panel ➤ 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.

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

Controls whether grips are automatically selected (made "hot") when subobjects are selected.

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.

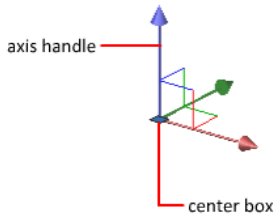
GTLOCATION

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.

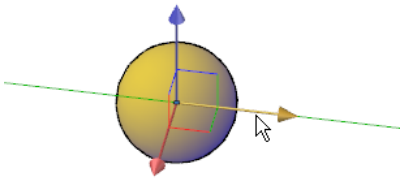
To move 3D objects and subobjects, click and drag the gizmo anywhere in 3D space. This location (indicated by the center box [or base grip] of the gizmo) sets the base point for the movement and temporarily changes the position of the UCS while you move the selected objects.



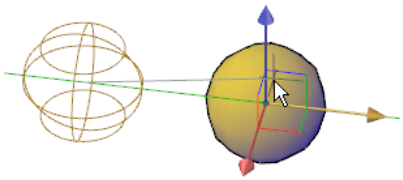
To move the objects freely, drag outside the gizmo or specify the axis or plane to which you will constrain the movement.

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.

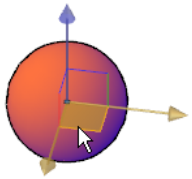


As you drag the cursor, movement of the selected objects and subobjects is constrained to the highlighted axis. You can click or enter a value to specify the distance of the move from the base point. If you enter a value, the movement direction of the object follows the initial direction of the cursor movement.

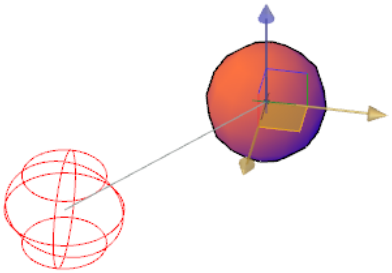


Constrain the Movement to a Plane

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 ➤ Move Gizmo. 
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.

   NOTE If a subobject filter is active, you do not need to press Ctrl to select a subobject. To select the entire object, turn off the filter.
3. When you have selected all objects, press Enter.

   The Move gizmo is displayed at the center of the selected objects.
4. Move the cursor over an axis handle on the gizmo until it turns yellow and the vector is displayed. Then click the axis handle.
5. Click another location or enter a value to specify the distance of the move.

To move objects in 3D space constrained to a specified plane

1. Click Home tab ➤ Modify panel ➤ Move Gizmo. 
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.

NOTE If a subobject filter is active, you do not need to press Ctrl to select a subobject. To select the entire object, turn off the filter.

3. When you have selected all objects, press Enter.

   The Move gizmo is displayed at the center of the selected objects.

4. 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.

5. Click a different location or enter a value to specify the distance of the move.

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

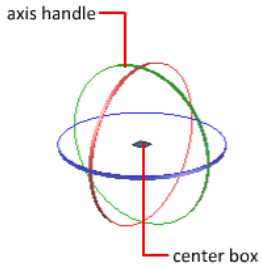
DEFAULTGIZMO

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.

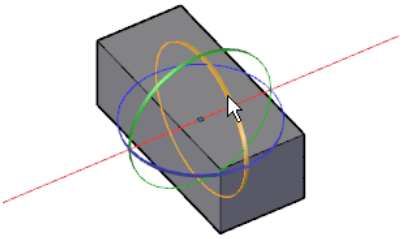


You then rotate the objects freely by dragging outside the gizmo. You can also specify an axis about which to constrain the rotation.

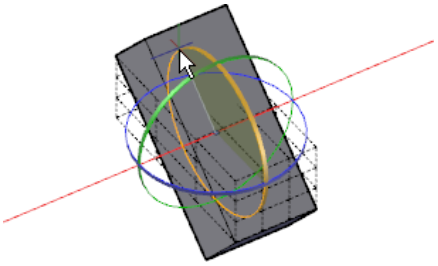
If you want to realign the center of rotation, you can relocate the gizmo by using the Relocate Gizmo option on the shortcut menu.

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 ➤ Rotate Gizmo. 
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.

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

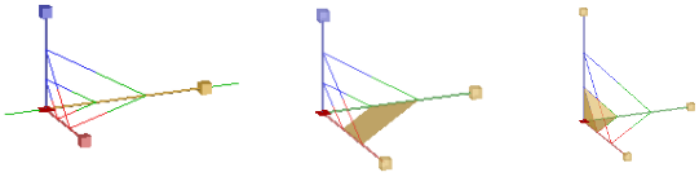
DEFAULTGIZMO

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.

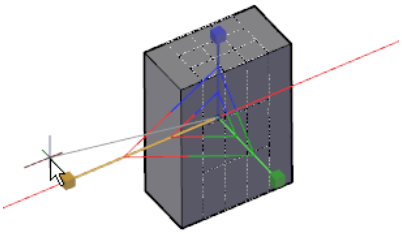
After you select the objects and subobjects to scale, constrain the object scaling by clicking the gizmo axis, plane, or the portion of the gizmo between all three axes.



NOTE Non-uniform scaling (along an axis or a plane) is only available for meshes, it is not available for solids and surfaces.

Scale a 3D Object Along an Axis

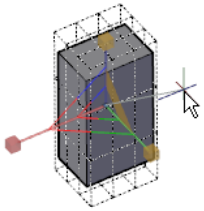
Constrain mesh object scaling to a specified axis. As you move the cursor over the axes on the 3D Scale gizmo, a vector line representing the axis of scale is displayed. Specify an axis of scale by clicking the axis when it turns yellow.



When you drag the cursor, the selected objects and subobjects are resized along the specified axis. Click or enter a value to specify the scale from the selected base point.

Scale a 3D Object Along a Plane

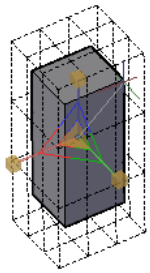
Constrain the mesh object scaling to a specified plane. Each plane is identified by a bar that extends from the outer ends of the respective axis handles. Specify the plane of scale by moving the cursor over one of the bars. When the bar turns yellow, click it.



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

Scale solid, surface, and mesh objects 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 ➤ Scale Gizmo. 
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.

To scale a 3D object along a specified plane

1. Click Home tab ➤ Modify ➤ Scale Gizmo. 
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.

To scale a 3D object uniformly

1. Click Home tab ➤ Modify ➤ Scale Gizmo. 
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.

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.

Use Grips to Modify Solids and Surfaces

Use grips to change the shape and size of solids and surfaces.

Use 3D Subobject Grips

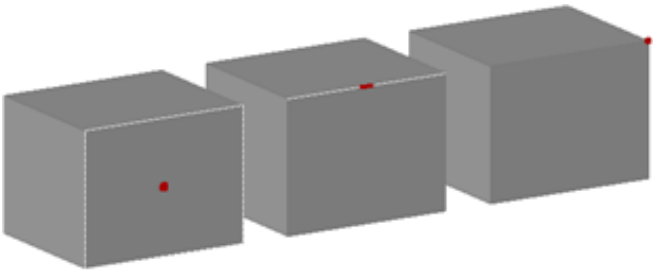
Select faces, edges, and vertices on 3D objects.

A *subobject* is a face, edge or vertex of a solid, surface, or mesh object.

Select Subobjects

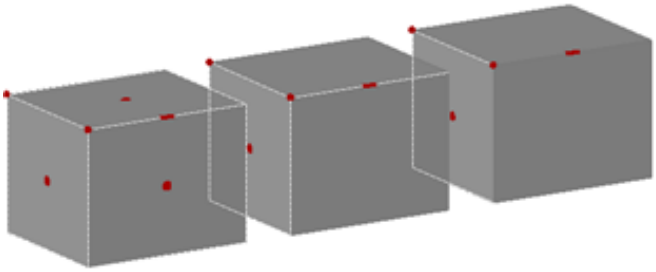
To select a face, edge, or vertex of a 3D object, press and hold Ctrl as you select the object. (If you have set a subobject filter, you do not need to hold Ctrl first.)

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 and hold Ctrl

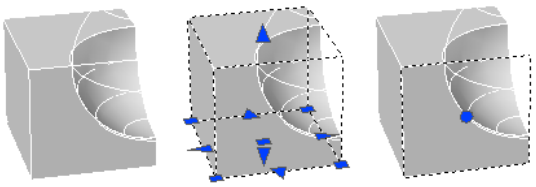
to select subobjects at the selection prompts of the MOVE, ROTATE, SCALE, and ERASE commands.



You can remove an item from the selection set by pressing and holding Shift 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 (On), 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 Ctrl and pick again to select a face, edge, or vertex on the original form.



If you set a subobject selection filter on page 1179, you can select the face, edge, or vertex by clicking it once.

To select one or more faces, edges, or vertices on a 3D object

1. Press Ctrl+click a face, edge, or vertex.

   NOTE If a subobject selection filter is active, you do not need to press Ctrl before clicking a face, edge, or vertex.

2. Repeat step 1 until all subobjects are selected.

To remove a subobject from a selection set

- Press and hold Ctrl+Shift. Click a selected face, edge, or vertex. The grip, if displayed, changes from red to blue.

NOTE If a subobject selection filter is active, you do not need to press Ctrl before clicking a face, edge, or vertex.

To select a face, edge, or vertex on the history form of a composite solid

1. Click Home tab ➤ Subobject panel ➤ Edge, Vertex, or Face.
2. On the history form, click a face, edge, or vertex.

NOTE If the Show History property for the history subobject is not turned on, you can display the history by pressing Ctrl as you move the cursor over the object.

Quick Reference

Commands

DSETTINGS

Sets grid and snap, polar and object snap tracking, object snap modes, Dynamic Input, and Quick Properties.

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 display of grips on selected objects.

GRIPSIZE

Sets the size of the grip box in pixels.

GRIPSUBOBJMODE

Controls whether grips are automatically selected (made "hot") when subobjects are selected.

LEGACYCTRLPICK

SUBOBJSELECTIONMODE

Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

Cycle Through and Filter Subobjects

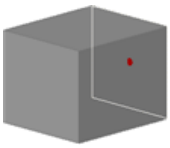
Filter and select faces, edges, and vertices on 3D objects.

A *subobject* is a face, edge or vertex of a solid, surface, or mesh object.

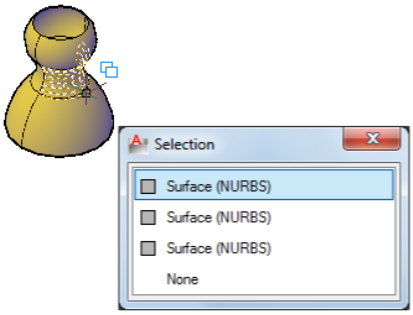
Cycle Through Multiple Subobjects

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 Cycling is turned on in the Selection Cycling Tab (Draft Settings Dialog Box).



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.



When a subobject selection filter is on, you do not need to press and hold Ctrl to select the face, edge, or vertex of a 3D model. However, you need to turn off the filter to select the entire object. The current subobject filter setting is stored in the SUBOBJSELECTIONMODE system variable.

When a subobject filter is turned on, the following icons are displayed near the cursor:

	Vertex filtering is on	Shift + F2
	Edge filtering is on	Shift + F3

	Face filtering is on	Shift + F4
	History subobject filtering is on	Shift + F5
	Subobject filtering is off	Shift + F1

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. Click a 3D solid or mesh subobject.

   Only the subobject type that you specified can be selected.

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

CULLINGOBJ

Controls whether 3D subobjects that are hidden from view can be highlighted or selected.

CULLINGOBJSELECTION

Controls whether 3D objects that are hidden from view can be highlighted or selected.

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 display of grips on selected objects.

GRIPSIZE

Sets the size of the grip box in pixels.

GRIPSUBOBJMODE

Controls whether grips are automatically selected (made “hot”) when subobjects are selected.

LEGACYCTRLPICK

Specifies the keys for selection cycling and the behavior for Ctrl+click.

SUBOBJSELECTIONMODE

Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

Use Grips to Edit 3D Solids and Surfaces

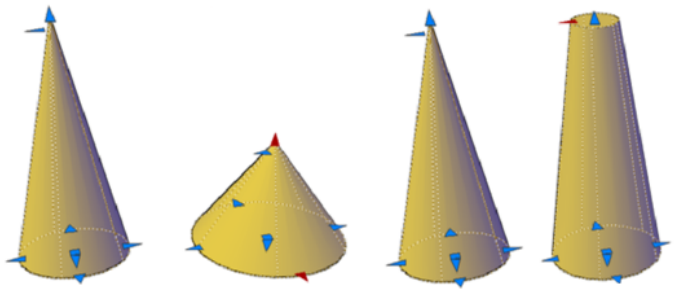
Use grips to change the size and shape of some individual solids and surfaces.

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.



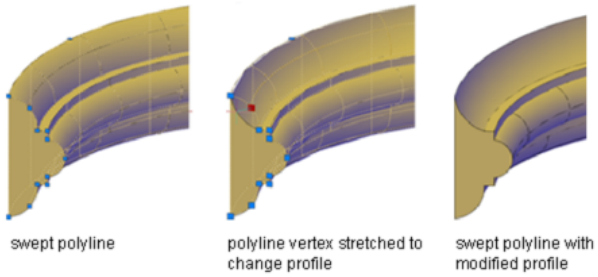
Extruded Solids and Surfaces

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.



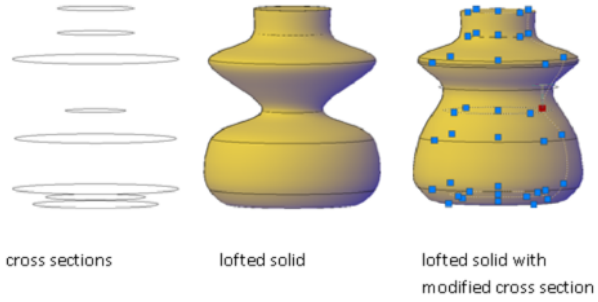
When you click and drag a grip on the profile, the changes are constrained to the plane of the profile curve.

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.

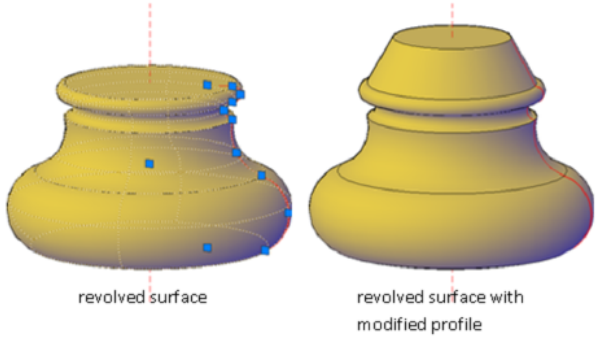


You cannot use grips to modify lofted solids or surfaces that are created with guide curves.

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 Solids on page 1049
- Create a Solid Cone on page 1056

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

Creates a 3D solid or surface by extending the dimensions of an object.

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 display of grips on selected objects.

GRIPSIZE

Sets the size of the grip box in pixels.

GRIPSUBOBJMODE

Controls whether grips are automatically selected (made "hot") when subobjects are selected.

Modify 3D Subobjects

Modify the shape of a 3D solid or surface by editing its subobjects (faces, edges, and vertices).

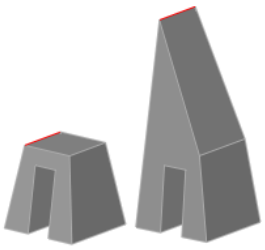
Move, Rotate, and Scale 3D Subobjects

Move, rotate, and scale individual subobjects on 3D solids and surfaces.

Use the same methods to modify a face, edge, or vertex that you use to modify the entire object:

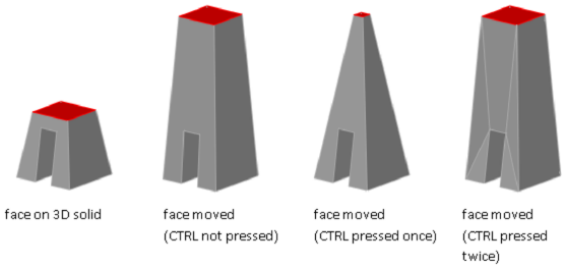
- Drag grips
- Use gizmos (3DMOVE, 3DROTATE, and 3DSCALE)
- Enter object editing commands (MOVE, ROTATE, and SCALE)

When you move, rotate, or scale a subobject, the subobject is modified in a way that maintains the integrity of the 3D solid or surface. 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 or surface. 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 (On).
- To modify subobjects of the combined composite solid as a whole, the History property must be set to None (Off).

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 1238

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.

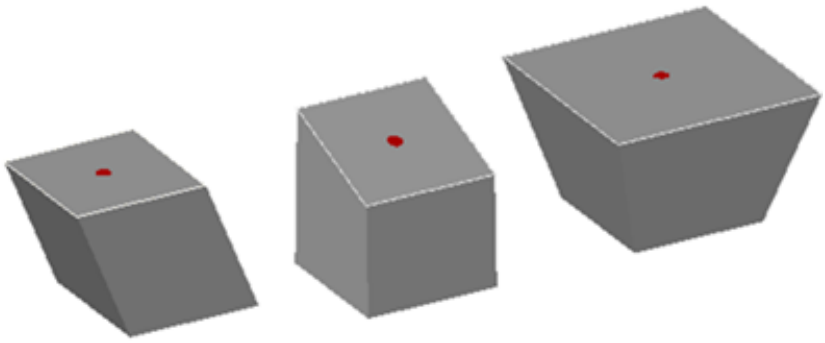
Modify Faces on 3D Objects

Modify individual faces on 3D objects using a variety of methods.

Move, Rotate, and Scale Faces on 3D Solids and Surfaces

You can select and modify faces of 3D solids and surfaces.

Modify the location, rotation, and size of faces on a 3D solids and surfaces.



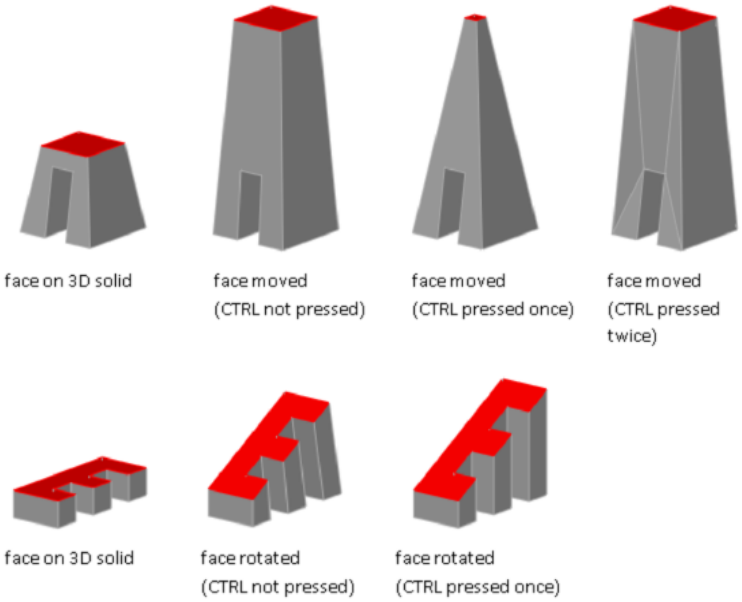
cube with top face moved, rotated, and scaled

Use the MOVE, ROTATE, and SCALE commands to modify faces just as you would with any other object. Press and hold Ctrl while you 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.



- **Extend 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.
- **Move Face**.When you move a face and press and release Ctrl once while dragging, the position of the face is modified within the boundary, or footprint, of the adjacent faces.
- **Allow Triangulation**.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.

TIP Rather than pressing Ctrl to cycle through face modification options, hover over a face grip to display the grip multi-functional menu.

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

GRIPMULTIFUNCTIONAL

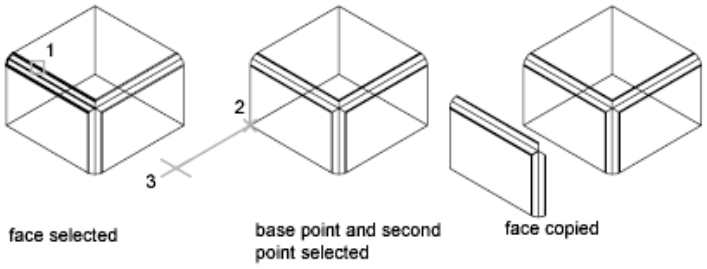
Specifies the access methods for multi-functional grip options.

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.

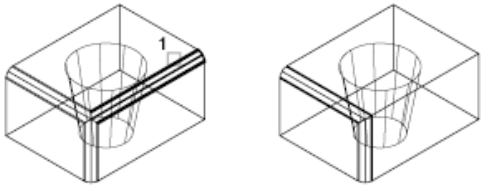


Delete a Face

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:

- Modify Mesh Faces on page 1253

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.

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.

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.

Quick Reference

Commands

PROPERTIES

Controls properties of existing objects.

SOLIDEDIT

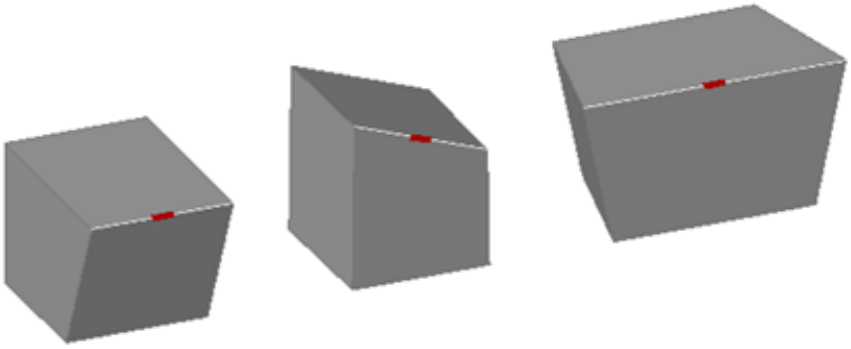
Edits faces and edges of 3D solid objects.

Modify Edges on 3D Objects

You can select and modify edges on a 3D solid or surface.

Move, Rotate, and Scale Edges

Move, rotate, and scale the edges on 3D solids and surfaces using grips, gizmos, and commands.



cubes with edges moved, rotated, and scaled

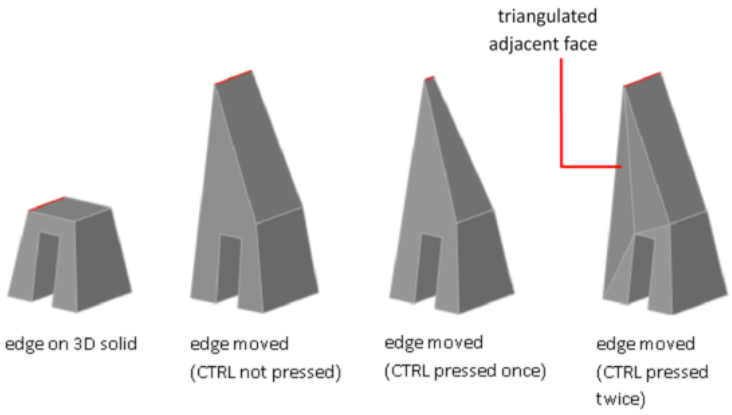
You can use MOVE, ROTATE, and SCALE to modify edges on 3D solids and surfaces just as you can for any other object. Press and hold Ctrl to select the edge.

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.



- **Extend Adjacent Faces**. 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.
- **Move 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.
- **Allow Triangulation**. 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.

TIP Rather than pressing Ctrl to cycle through edge modification options, hover over an edge grip to display the grip multi-functional menu.

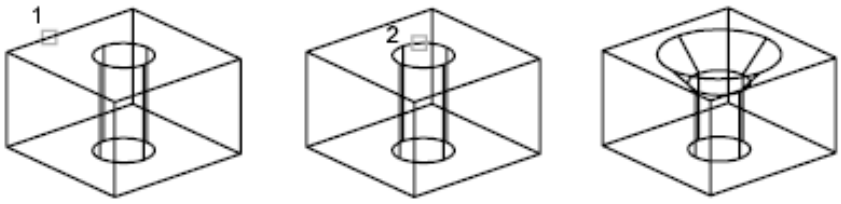
Delete Edges

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

Round, fillet, or bevel the edges of 3D solids using FILLETEDGE and CHAMFEREDGE.



Use the fillet and chamfer grips to modify the fillet radius or the chamfer distance. The default fillet radius is set by the FILLETRAD3D system variable.

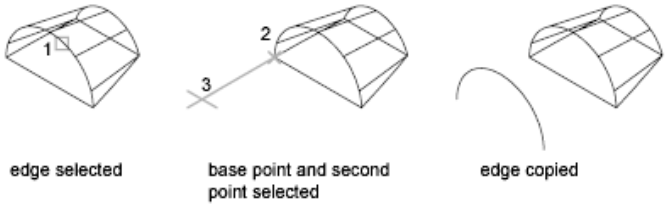
TIP Use the *Chain* option of FILLETEDGE to limit selection to tangent edges.

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 1186

- Use Gizmos to Modify Objects on page 1161
- Modify Objects on page 775
- Move or Rotate Objects on page 787
- Resize or Reshape Objects on page 828
- Overview of Modifying Meshes on page 1238

To fillet a solid object



1. Click Solid tab ➤ Solid Editing panel ➤ Fillet Edge.
2. Select the edge of the solid to fillet.
3. Specify the fillet radius.
4. Select additional edges or press Enter to fillet.

To chamfer a 3D solid object



1. Click Solid tab ➤ Solid Editing panel ➤ Chamfer Edge.
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.

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.

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.

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.

Quick Reference

Commands

EDGE

Changes the visibility of 3D face edges.

CHAMFER

Bevels the edges of objects.

CHAMFEREDGE

Bevels the edges of 3D solids and surfaces.

FILLET

Rounds and fillets the edges of objects.

FILLETEDGE

Rounds and fillets the edges of solid 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

FILLETRAD3D

Stores the current fillet radius for 3D objects.

GRIPMULTIFUNCTIONAL

Specifies the access methods for multi-functional grip options.

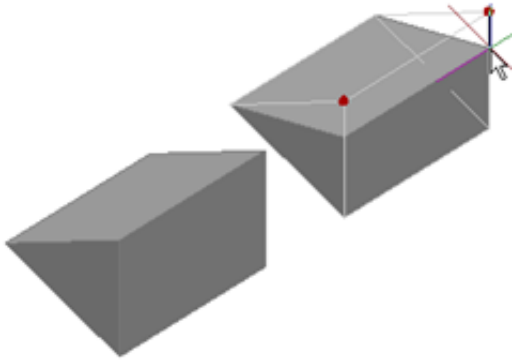
PREVIEWCREATIONTRANSPARENCY

Controls the transparency of the preview generated while using SURFBLEND, SURFPATCH, SURFFILLET, FILLETEDGE, CHAMFEREDGE, and LOFT.

Modify Vertices on 3D Objects

You can select and modify 3D object vertices.

Move, rotate, scale, or drag the vertices of 3D solids and surfaces.



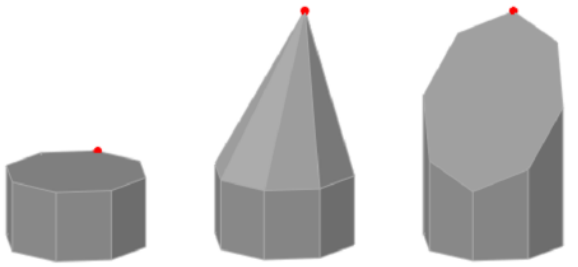
wedge with two vertices moved

You can modify the form of a 3D solid or surface 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 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 Properties palette.

Vertex Modification Options

As you drag a vertex, press Ctrl to cycle through modification options.



- **Move Vertex**. When you move a vertex and press and release Ctrl once, adjacent planar faces might be adjusted.
- **Allow Triangulation**. When you move a vertex without pressing Ctrl, some adjacent planar faces may be triangulated (divided into two or more planar triangular faces).

If you press and release Ctrl a second time, the modification returns to the first option, as if you had not pressed Ctrl.

TIP Rather than pressing Ctrl to cycle through vertex modification options, hover over a vertex grip to display the grip multi-functional menu.

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 1186
- Use Gizmos to Modify Objects on page 1161
- Modify Objects on page 775
- Overview of Modifying Meshes on page 1238

To move a vertex on a 3D object

1 Click Home tab ➤ Subobject panel ➤ Vertex. 

2 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 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

GRIPMULTIFUNCTIONAL

Specifies the access methods for multi-functional grip options.

Work with Complex 3D Solids and Surfaces

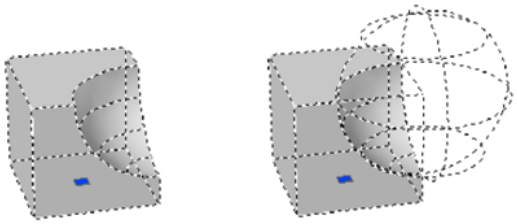
Modify composite solids that are created by a union, subtract, intersect, fillet, or chamfer process.

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 (On), 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

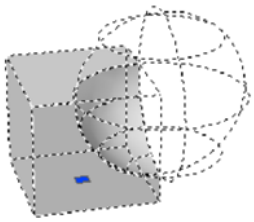
composite solid with Show History set to yes

To retain a history of the original parts of composite solids, the History property must be set to Record (On) in the Properties palette when the composite operation occurs. You can also use the SOLIDHIST system variable to set this property.

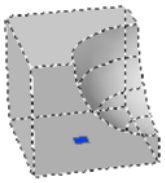
Display and Remove the History to Modify the Composite

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 1205.

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 (Off) to remove the history. Then reset the value to Record (On). With this process, you can create a complex composite object, and then reset it to serve as a base form for additional composite operations.

See also:

- Modify Composite Solids and Surfaces on page 1205

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.

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 solids and composite solids, and associativity from surfaces.

System Variables

SHOWHIST

Controls the Show History property for solids in a drawing.

SOLIDHIST

Controls whether new composite solids retain a history of their original components.

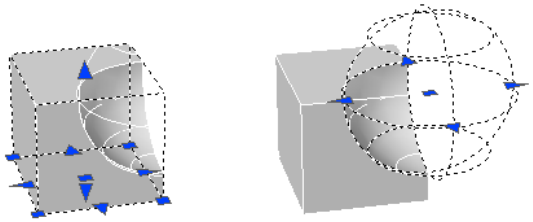
Modify Composite Solids and Surfaces

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

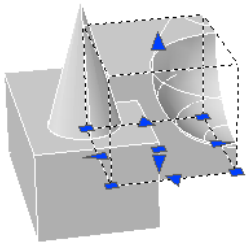
When the History property is set to Record (On), press the Ctrl key to display any original forms that were removed during a union, subtract, or intersect operation. If the original, removed form was a solid primitive, you can drag the displayed grips to change its shape and size. As a result, the composite 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 Subobjects on page 1186.

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.) Composite objects cannot overlap or share a common area or volume to be separated.

After separation, the individual solids retain their original layers and colors. All nested 3D solid objects are restored to their simplest forms.

See also:

- Modify Objects Using Grips on page 777

- Overview of Modifying Meshes on page 1238

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.

Quick Reference

Commands

SOLIDEDIT

Edits faces and edges of 3D solid objects.

System Variables

LEGACYCTRLPICK

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.

Quick Reference

Commands

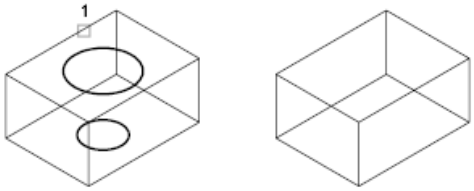
SOLIDEDIT

Edits faces and edges of 3D solid objects.

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 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.

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.

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.



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.

Quick Reference

Commands

EXTRUDE

Creates a 3D solid or surface by extending the dimensions of an object.

PRESSPULL

Presses or pulls bounded areas.

System Variables

IMPLIEDFACE

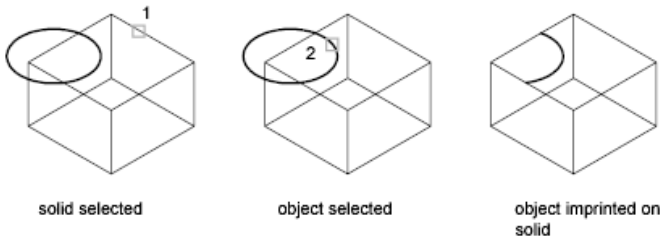
Controls the detection of implied faces.

Add Facets to Faces on Solids and Surfaces

Subdivide faces into additional facets on 3D solids and surfaces by imprinting other objects, such as arcs and circles on them.

With the IMPRINT command, you can add a new facet to a 3D solid by imprinting a coplanar object that overlaps a face. Imprinting provides additional edges that you can use to reshape the 3D object.

For example, if a circle overlaps the face of a box, you can imprint it on the solid.



You can delete or retain the original object when you imprint it.

Objects that can be imprinted on 3D solids 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 drag it to a new location, or you can use PRESSPULL on the facet.

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.

Quick Reference

Commands

IMPRINT

Imprints 2D geometry on a 3D solid or surface, creating additional edges on planar faces.

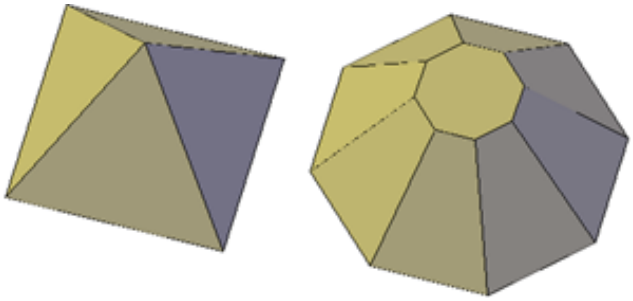
Modify the Properties of 3D Solids, Surfaces, and Meshes

Modify 3D objects by changing their settings in the Properties palette.

3D solids, surfaces, and meshes, and their subobjects can be modified in the Properties palette.

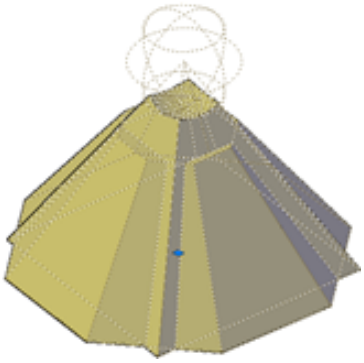
Modify Solid Objects by Changing Properties

By changing settings in the Properties palette, you can modify basic size, height, and shape characteristics of primitive solids. 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. For more information, see Work with Complex 3D Solids and Surfaces on page 1202.



Modify Surface Objects by Changing Properties

Surface objects have additional properties that are not found in 3D solid or mesh objects. The properties differ depending on the type of surface (NURBS, blend, patch, network, offset, fillet, chamfer, extend, loft, extrude, sweep, planar, or revolve).

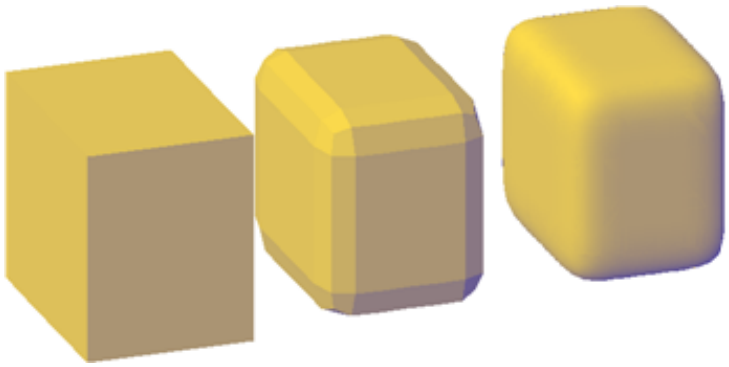
Surfaces include the following information in the Properties palette:

- **Basic geometric information** - Contains information such as radius for fillet surfaces, offset distance for offset surfaces, and taper angle for extruded surfaces. You can also enter mathematical expressions to control some of these properties.
- **Maintain Associativity** - Displays whether surface is associative or not. Use this property to turn associativity off.
- **Show Associativity** - Turns dependency highlighting on and off if the surface is associated with other surfaces.
- **Edge Continuity and Bulge Magnitude** - Displays for surfaces that join other surfaces.
- **Wireframe Display and U/V Isolines** - Turns the wireframe and U/V Isoline display on and off (for non-NURBS surfaces).
- **CV Hull Display and U/V Isoparms** - Turns the CV Hull and U/V Isoparm display on and off (for NURBS surfaces).
- **Trims** - Reports whether the surface has any trimmed areas and on which edges.

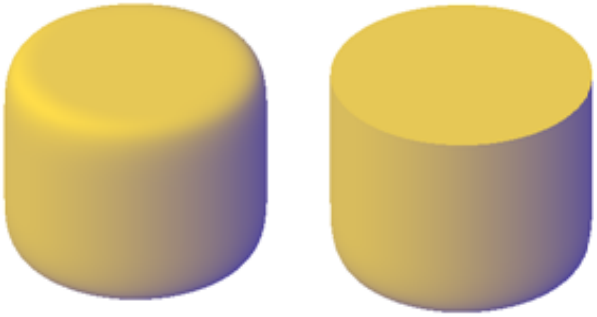
Modify Mesh Objects by Changing Properties

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.

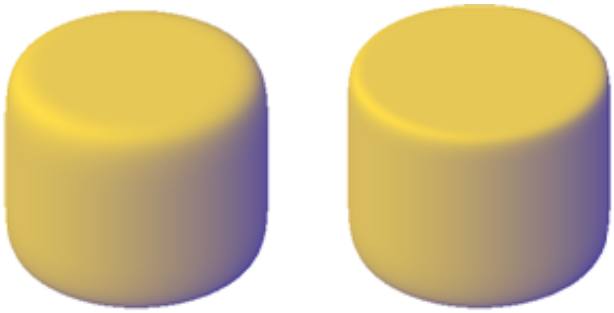
- **Level of Smoothness.** Smooths or sharpens the edges of a mesh object.



- **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.



Modify 3D Subobject Properties

In addition to solids, surfaces, and meshes, you can also use the Properties palette to modify the 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 New Materials on page 2209.

See also:

- Work with Complex 3D Solids and Surfaces on page 1202

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 Loft Settings.)

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.

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.

Quick Reference

Commands

PROPERTIES

Controls properties of existing objects.

System Variables

FILLETRAD

Stores the current fillet radius for 2D objects.

FILLETRAD3D

Stores the current fillet radius for 3D objects.

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.

Modify Surfaces

Reshape surfaces, and then analyze and rebuild the model if necessary to ensure quality and smoothness.

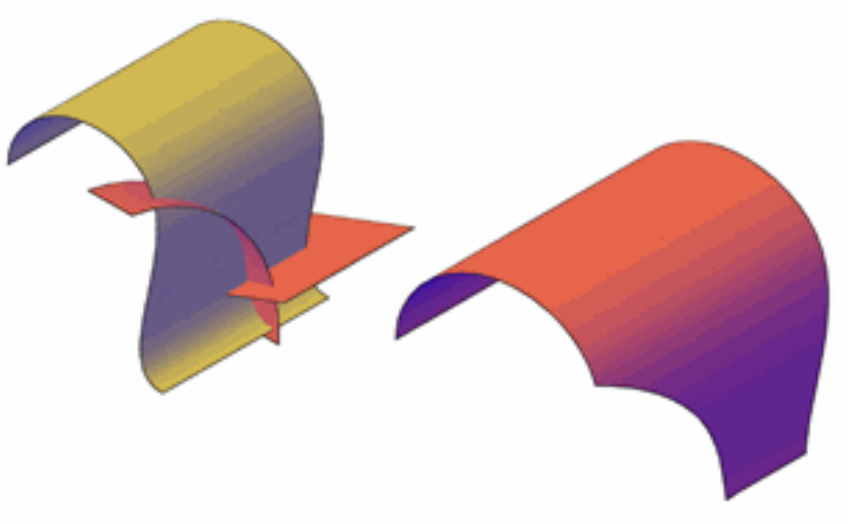
Overview of Modifying Surfaces

Fillet, extend, and trim surfaces or modify NURBS surfaces with control vertices.

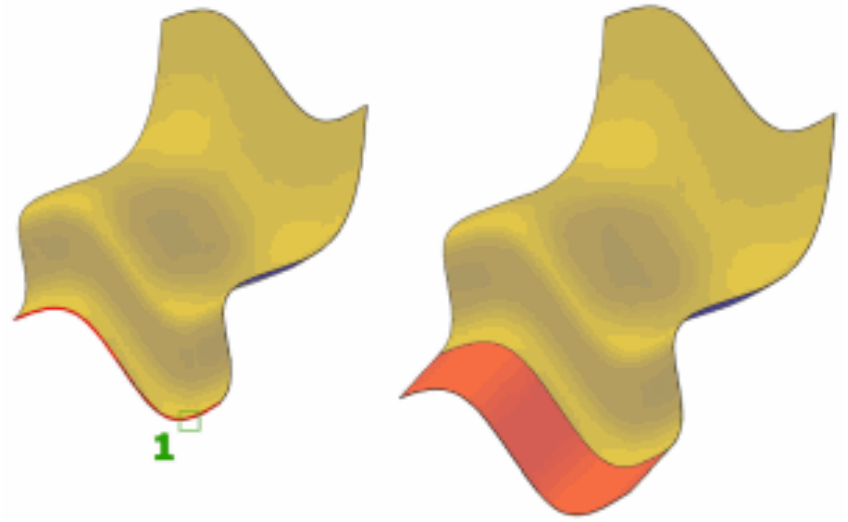
Procedural and NURBS surfaces can be edited with basic editing tools such as trimming, extending and filleting. NURBS surfaces can be modified with these tools as well, but they can also be reshaped by stretching control vertices.

When your surface design is complete, use the surface analysis tools to ensure the quality of your model and to rebuild it if necessary.

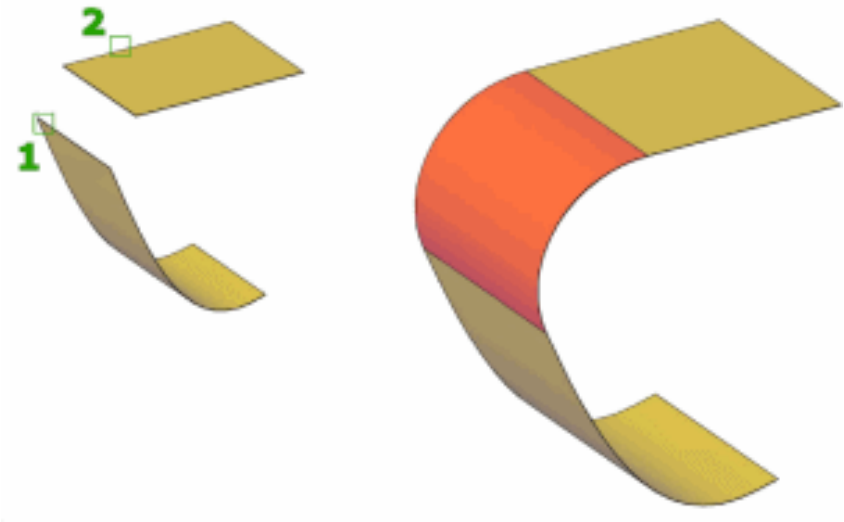
- **Trim and Untrim Surfaces**



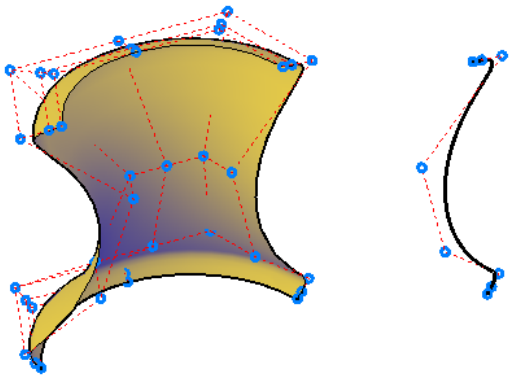
- **Extend Surfaces**



- **Fillet Surfaces**



■ **Edit NURBS Surfaces**



Trim and Untrim Surfaces

Trim and untrim surfaces to meet the edges of other objects.

An important step in the surface modeling workflow is trimming surfaces. You can trim a surface where it meets an intersecting object or you can project geometry onto a surface as a trimming edge.

When a surface is trimmed, the removed surface areas can be replaced with SURFUNTRIM.

NOTE SURFUNTRIM does not restore areas removed by the SURFAUTOTRIM system variable and PROJECTGEOMETRY. It only restores areas trimmed with SURFTRIM.

The Properties palette indicates if the surface contains any trimmed edges.

Projecting Geometry onto Surfaces, Solids, and Regions

Similar to projecting a movie onto a screen, you can project geometry onto 3D solids, surfaces, and regions from different directions to create trimming edges. The PROJECTGEOMETRY command creates a duplicate curve on the object that you can move and edit. You can also trim against 2D curves that do not actually touch the surface, but that appear to intersect the object in the current view.

Use the SURFACEAUTOTRIM system variable to automatically trim a surface when you project geometry onto it.

Options for Projecting Geometry

Project geometry from 3 different angles: the Z axis of the current UCS, the current view, or a path between two points.

- **Project to UCS** - Projects the geometry along the positive or negative Z axis of the current UCS.
- **Project to View** - Projects the geometry based on the current view.
- **Project to Two Points** - Projects the geometry along a path between two points.

To trim a surface

1. Click Surface tab ➤ Create panel ➤ Trim. 
2. Select the surface and press Enter.
3. Select the cutting curves and press Enter.
4. Select the surface areas to remove and press Enter.
   The selected areas are removed.

To untrim a surface

1. Click Surface tab ➤ Create panel ➤ Untrim. 
2. Select a visible part of the surface and press Enter.
   The trimmed areas are replaced.

NOTE When you trim a surface multiple times, you may lose some of the original trimming edges. In this case, some untrim actions may not be possible if the trimming edge has been lost.

To automatically trim areas to projected curves

1. Click Surface tab ➤ Project Geometry panel ➤ Auto Trim. 
2. Click Surface tab ➤ Project Geometry panel ➤ Project to View. 
3. Select the curve to project onto the surface and press Enter.
4. Select a surface as the target and press enter.
   The area is automatically trimmed.

Quick Reference

Commands

3DOSNAP

Sets the object snap modes for 3D objects.

PROJECTGEOMETRY

Projects points, lines, or curves onto a 3D solid or surface from different directions.

PROPERTIES

Controls properties of existing objects.

SURFTRIM

Trims portions of a surface where it meets another surface or type of geometry.

SURFUNTRIM

Replaces surface areas removed by the SURFTRIM command.

VISUALSTYLES

Creates and modifies visual styles and applies a visual style to a viewport.

System Variables

3DOSMODE

Controls the settings for the 3D object snaps.

SUBOBJSELECTIONMODE

Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

SURFACEAUTOTRIM

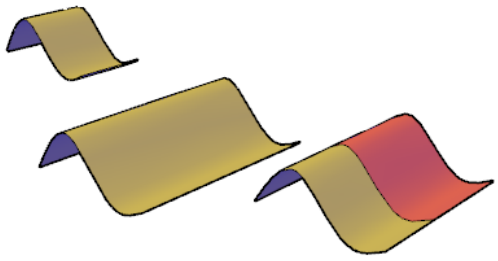
Controls whether surfaces are automatically trimmed when you project geometry onto them.

Extend a Surface

Create a new surface by extending it to meet the edge of another object or by specifying an extension length.

There are two types of extend surfaces: merge and append. The *merge* surface is a continuation of the surface with no seam. The *append* surface extends the surface by adding a second surface with a seam. Because it creates a seam, append surfaces have continuity and bulge magnitude on page 1082 properties.

For both surface types, use the Properties palette to change the length or to derive the length from a mathematical expression.



To create a new surface by extending a surface

1. Select a surface.
2. Right-click and select Properties.
3. In the Properties palette, under Geometry, change the desired settings.

Quick Reference

Commands

PROPERTIES

Controls properties of existing objects.

SURFEXTEND

Lengthens a surface by a specified distance.

Fillet a Surface

Create a new transition surface that fillets an area between two existing surfaces or regions.

Create a tangent surface between two surfaces or regions with a constant radius profile. The original surfaces will trim to meet the fillet surface.

By default, the fillet surface uses the radius value set in the FILLETRAD3D system variable. Change the radius while you are creating the surface with the radius option or by dragging the fillet grip. Use the Properties palette to change the fillet radius or to derive the radius using a mathematical expression.

To create a fillet surface

1. Click Surface tab ➤ Edit panel ➤ Fillet. 
2. Select the first and second surfaces.

   The fillet surface is created and the originating are surfaces automatically trimmed if necessary.

Quick Reference

Commands

SURFFILLET

Creates a filleted surface between two other surfaces.

PROPERTIES

Controls properties of existing objects.

System Variables

FILLETRAD3D

Stores the current fillet radius for 3D objects.

PREVIEWCREATIONTRANSPARENCY

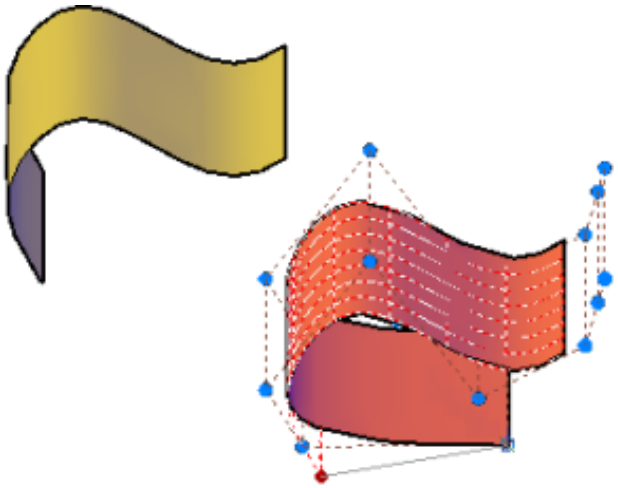
Controls the transparency of the preview generated while using SURFBLEND, SURFPATCH, SURFFILLET, FILLETEDGE, CHAMFEREDGE, and LOFT.

Edit NURBS Surfaces

Change the shape of NURBS surfaces and curves by using the 3D Edit Bar or by editing control vertices.

Use the Control Vertices Edit Bar (3DEDITBAR) to drag and reshape surfaces.

A second way to edit NURBS surfaces is to drag and edit the control vertices directly. Press and hold Shift to select multiple control vertices.



Use CVSHOW to display the control vertices for both NURBS surfaces and curves.

Drag the control vertices to reshape the curve or surface; you can also add or delete control vertices in both the *U* and *V* directions.

The typical surface modeling workflow is to:

- Create a model that combines 3D solids, surfaces, and mesh objects.
- Convert the model to procedural surfaces to take advantage of associative modeling.
- Convert the procedural surfaces to NURBS surfaces with CONVTONURBS to take advantage of NURBS editing.
- Check for imperfections and wrinkles with the surface analysis tools.
- If necessary, rebuild the surfaces to restore smoothness with CVREBUILD.

Quick Reference

Commands

3DEDITBAR

CONVTONURBS

Converts 3D solids and surfaces into NURBS surfaces.

CVSHOW

Displays the control vertices for specified NURBS surfaces or curves.

CVADD

Adds control vertices to NURBS surfaces and splines.

CVHIDE

Turns off the display of control vertices for all NURBS surfaces and curves.

CVREBUILD

Rebuilds the shape of NURBS surfaces and curves.

CVREMOVE

Removes control vertices from NURBS surfaces and curves.

System Variables

SURFACEMODELINGMODE

Controls whether surfaces are created as procedural surfaces or NURBS surfaces.

Rebuild NURBS Surfaces and Curves

Reconstruct NURBS surfaces and curves to a specified degree and number of control vertices.

Editing a NURBS surface or curve can create discontinuity and wrinkles. Reconstruct the surface or curve by changing the degree and the number of control vertices. Rebuilding also allows you to delete the original geometry, and, for surfaces only, to replace trimmed areas.

To rebuild a NURBS surface

1. Click Surface tab ➤ Control Vertices panel ➤ Rebuild. 
   The Rebuild Surface dialog box displays.
2. Select the desired rebuild options and click OK.

Quick Reference

Commands

CONVTONURBS

Converts 3D solids and surfaces into NURBS surfaces.

CVSHOW

Displays the control vertices for specified NURBS surfaces or curves.

CVADD

Adds control vertices to NURBS surfaces and splines.

CVHIDE

Turns off the display of control vertices for all NURBS surfaces and curves.

CVREBUILD

Rebuilds the shape of NURBS surfaces and curves.

CVREMOVE

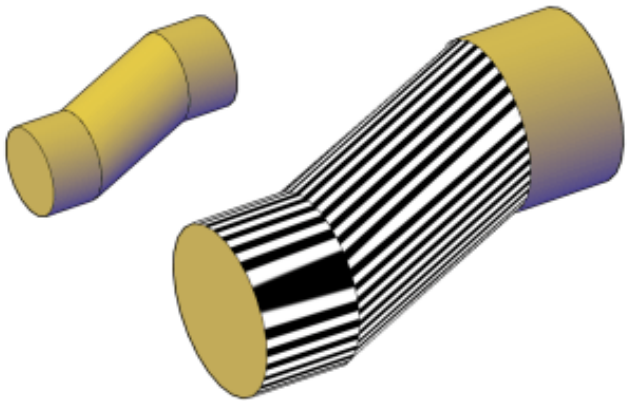
Removes control vertices from NURBS surfaces and curves.

Analyze Surfaces

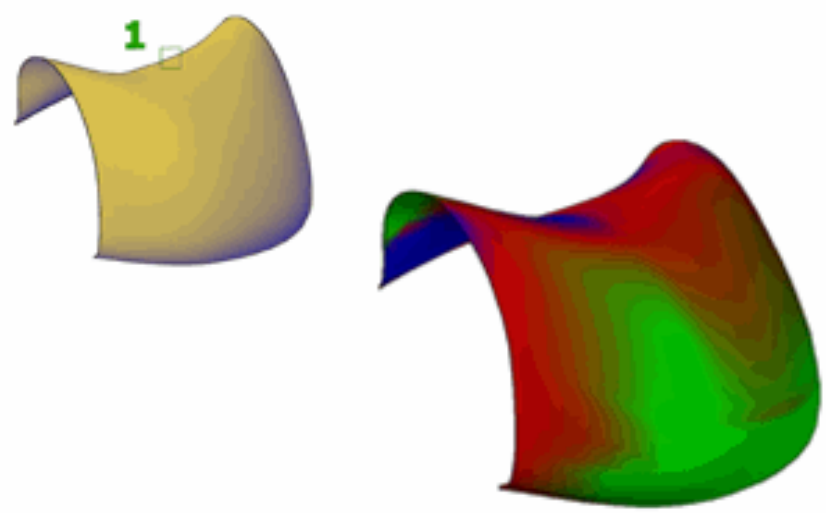
Surface analysis tools check the continuity, curvature and draft angles of surfaces.

Use the surface analysis tools to validate surfaces and curves before manufacturing. Analysis tools include:

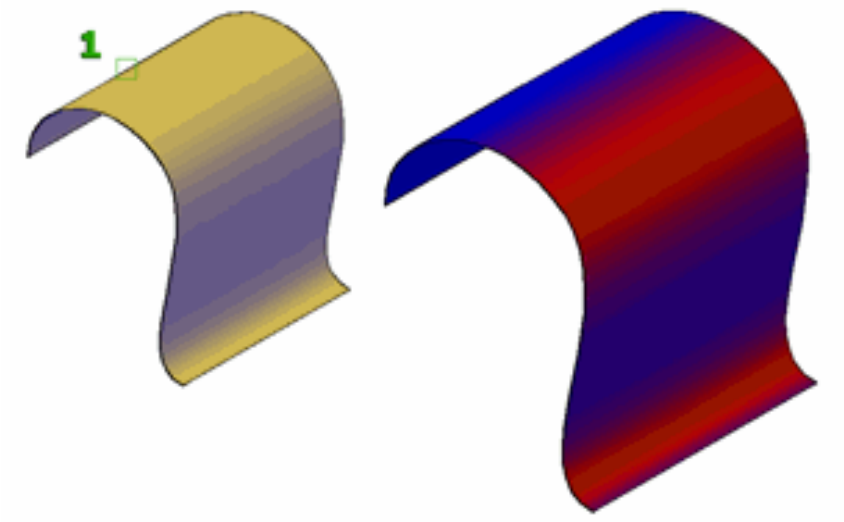
- Zebra Analysis on page 1232 - Analyzes surface continuity by projecting parallel lines onto the model.



- Curvature Analysis on page 1235 - Evaluates areas of high and low surface curvature by displaying a color gradient.



- Draft Analysis on page 1236 - Evaluates whether a model has adequate draft between a part and its mold.



NOTE Analysis tools only work in the 3D visual styles; they will not work in 2D.

Quick Reference

Commands

ANALYSISCURVATURE

Displays a color gradient onto a surface to evaluate different aspects of its curvature.

ANALYSISDRAFT

Displays a color gradient onto a 3D model to evaluate whether there is adequate space between a part and its mold.

ANALYSISOPTIONS

Sets the display options for zebra, curvature, and draft analysis.

ANALYSISZEBRA

Projects stripes onto a 3D model to analyze surface continuity.

System Variables

VSACURVATUREHIGH

Sets the value at which a surface displays as green during curvature analysis (ANALYSISCURVATURE).

VSACURVATURELOW

Sets the value at which a surface displays as blue during curvature analysis (ANALYSISCURVATURE).

VSACURVATURETYPE

Controls which type of curvature analysis is used with the (ANALYSISCURVATURE).

VSADRAFTANGLEHIGH

Sets the value at which a model displays as green during draft analysis (ANALYSISDRAFT).

VSADRAFTANGLELOW

Sets the value at which a model displays as blue during draft analysis (ANALYSISDRAFT).

VSAZEBRACOLOR1

Sets the first color of the zebra stripes displayed during zebra analysis (ANALYSISZEBRA).

VSAZEBRACOLOR2

Sets the second (contrasting) color of the zebra stripes displayed during zebra analysis (ANALYSISZEBRA).

VSAZEBRADIRECTION

Controls whether zebra stripes display horizontally, vertically, or at an angle during zebra analysis (ANALYSISBRA).

VSAZEBRASIZE

Controls the width of the zebra stripes displayed during zebra analysis (ANALYSISZEBRA).

VSAZEBRATYPE

Sets the type of zebra display when using zebra analysis (ANALYSISZEBRA).

Analyze Surface Continuity with Zebra Analysis

The zebra analysis tool projects stripes onto a surface so that you can inspect the continuity between surfaces.

Surface continuity is a measure of how smoothly two surfaces flow into each other. A car hood, for example, can be composed of multiple small surfaces that appear to be one because of the smoothness of the surface continuity.

NOTE Analysis tools only work in the 3D visual styles; they will not work in 2D.

How to Interpret the Zebra Stripes

In the seam where two surfaces meet, the way that the zebra stripes align and curve tells you a lot about the smoothness of the join.

- **G0 Position.** The position of the surface edges is collocated; they touch. But the tangency and curvature do not match. The zebra stripes do not line up.



- **G1 Tangency.** The position and tangency of surfaces is the same. This indicates G1 (G0 + G1 or position + tangency). The zebra stripes line up, but they veer away from one another at sharp curves.



- **G2 Curvature.** The position, tangency, and curvature of the surface edges is the same. This indicates G2 (G0 + G1 + G2 or position + tangency + curvature).
  The stripes line up, but they do not veer away from each other at sharp curves (because they share the same curvature). This distinction is subtle and a little harder to discern from G1 continuity.



To analyze surface continuity

1. Click Surface tab ➤ Analysis panel ➤ Zebra. 
2. Select a surface and press Enter.

The zebra stripes display.

To turn off the zebra display

1. Click Surface tab ➤ Analysis panel ➤ Analysis Options. 
2. Click the **Clear Zebra Analysis** button.

The zebra stripes disappear.

To change the zebra analysis options

1. Click Surface tab ➤ Analysis panel ➤ Analysis Options. 
2. In the Zebra tab of the Analysis Options dialog box, select the desired options and click **OK**.

Quick Reference

Commands

ANALYSISOPTIONS

Sets the display options for zebra, curvature, and draft analysis.

ANALYSISZEBRA

Projects stripes onto a 3D model to analyze surface continuity.

System Variables

VSAZEBRACOLOR1

Sets the first color of the zebra stripes displayed during zebra analysis (ANALYSISZEBRA).

VSAZEBRACOLOR2

Sets the second (contrasting) color of the zebra stripes displayed during zebra analysis (ANALYSISZEBRA).

VSAZEBRADIRECTION

Controls whether zebra stripes display horizontally, vertically, or at an angle during zebra analysis (ANALYSISBRA).

VSAZEBRASIZE

Controls the width of the zebra stripes displayed during zebra analysis (ANALYSISZEBRA).

VSAZEBRATYPE

Sets the type of zebra display when using zebra analysis (ANALYSISZEBRA).

Analyze the Curvature of a NURBS Surface

Displays a color gradient onto surfaces to evaluate areas of high, low and Gaussian curvature.

The color gradient allows you to visualize Gaussian, minimum, maximum, and mean *U* and *V* surface curvature. Maximum curvature and a positive Gaussian value display as red, and minimum curvature and a negative Gaussian value display as blue. Positive Gaussian curvature means the surface is shaped like a bowl. Negative Gaussian curvature means the surface is shaped like a saddle as shown in this illustration. Mean curvature and a zero Gaussian value means the surface is flat in at least one direction (planes, cylinders, and cones have zero Gaussian curvature).

The curvature analysis tool ensures that your model stays within a specified range. You can analyze the following on a point on a surface or curve:

NOTE Analysis tools only work in the 3D visual styles; they will not work in 2D.

To analyze the curvature of a surface

1. Click Surface tab ➤ Analysis panel ➤ Curvature. 
2. Select a surface and press Enter.

The curvature coloring display.

To turn off the curvature color display

1. Click Surface tab ➤ Analysis panel ➤ Analysis Options. 
2. Click the **Clear Curvature Analysis** button.

The curvature coloring disappears.

To change the curvature analysis options

1. Click Surface tab ➤ Analysis panel ➤ Analysis Options. 
2. In the Curvature tab of the Analysis Options dialog box, select the desired options and click **OK**.

Quick Reference

Commands

ANALYSISCURVATURE

Displays a color gradient onto a surface to evaluate different aspects of its curvature.

ANALYSISOPTIONS

Sets the display options for zebra, curvature, and draft analysis.

System Variables

VSACURVATUREHIGH

Sets the value at which a surface displays as green during curvature analysis (ANALYSISCURVATURE).

VSACURVATURELOW

Sets the value at which a surface displays as blue during curvature analysis (ANALYSISCURVATURE).

VSACURVATURETYPE

Controls which type of curvature analysis is used with the (ANALYSISCURVATURE).

Use the Draft Analysis Tool

Evaluates if a model has adequate space between a part and its mold.

If you are creating shapes or parts that need to be molded, the draft analysis tool evaluates if their is adequate draft between a part and its mold (based on the pull direction).

NOTE Analysis tools only work in the 3D visual styles; they will not work in 2D.

To analyze the draft of a model

1. Click Surface tab ➤ Analysis panel ➤ Draft. 
2. Select a model and press Enter.

The draft analysis colors display.

To turn off the draft color display

1. Click Surface tab ➤ Analysis panel ➤ Analysis Options. 
2. Click the **Clear Draft Analysis** button.

The draft analysis colors disappears.

To change the draft analysis options

1. Click Surface tab ➤ Analysis panel ➤ Analysis Options. 
2. In the Draft Angle tab of the Analysis Options dialog box, select the desired options and click **OK**.

Quick Reference

Commands

ANALYSISDRAFT

Displays a color gradient onto a 3D model to evaluate whether there is adequate space between a part and its mold.

ANALYSISOPTIONS

Sets the display options for zebra, curvature, and draft analysis.

System Variables

VSADRAFTANGLEHIGH

Sets the value at which a model displays as green during draft analysis (ANALYSISDRAFT).

VSADRAFTANGLELOW

Sets the value at which a model displays as blue during draft analysis (ANALYSISDRAFT).

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.

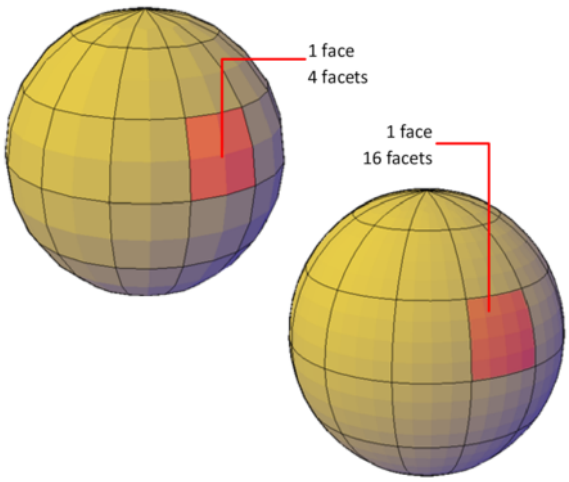
Mesh objects do not have the mass and volume properties of 3D solids. However, they do offer unique capabilities that enable you to design less angular, more rounded models. Mesh objects are easier to mold and reshape than their solid and surface counterparts.



NOTE The capabilities described in this section apply only to mesh objects created in 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 (MESHSMOOTHMORE, MESHSMOOTHLESS).
- **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 (MESHREFINE).

- **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 (MESHCREASE).
- **Split or merge faces.** Divide an existing face into separate components along a specified path. Merge two or more faces to create a single face (MESHSPLIT MESHMERGE).
- **Collapse vertices.** Alter the mesh model by collapsing the vertices of adjacent faces to a single point (MESHCOLLAPSE.
- **Spin edges.** Spin the shared edge of adjacent triangular faces to alter the shapes and orientation of the faces (MESHSPIN).
- **Extrude a face.** Extend a specified face by extruding it into 3D space. Unlike 3D solid extrusion, a mesh extrusion does not create a separate object (MESHEXTRUDE).
- **Repair holes.** Close a gap between faces by selecting the surrounding edges. Holes in mesh objects can prevent you from converting a mesh object to a solid object (MESHCAP).

Use Grip Editing with Mesh

Grips, as described in Use Grips to Edit 3D Solids and Surfaces on page 1183, 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 and hold Ctrl while selecting a subobject. The subobject highlighting indicates what is selected. Press and hold 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, which you can select without pressing and holding Ctrl. See Use 3D Subobject Grips on page 1176.
- **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. Form more information, see Use Gizmos to Modify Objects on page 1161.

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.

See also:

- Use Grips to Edit 3D Solids and Surfaces on page 1183
- Use Gizmos to Modify Objects on page 1161

Quick Reference

Commands

MESHCAP

Creates a mesh face that connects open edges.

MESHCOLLAPSE

Merges the vertices of selected mesh faces or edges.

MESHCREASE

Sharpens the edges of selected mesh subobjects.

MESHEXTRUDE

Extends a mesh face into 3D space.

MESHMERGE

Merges adjacent faces into a single face.

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.

MESHSPIN

Spins the adjoining edge of two triangular mesh faces.

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

Controls whether grips are automatically selected (made "hot") when subobjects are selected.

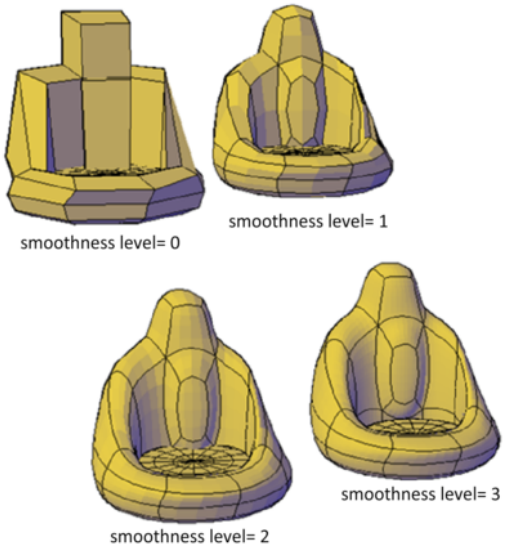
VSLIGHTINGQUALITY

Sets the lighting quality in the current viewport.

Change Mesh Smoothness Levels

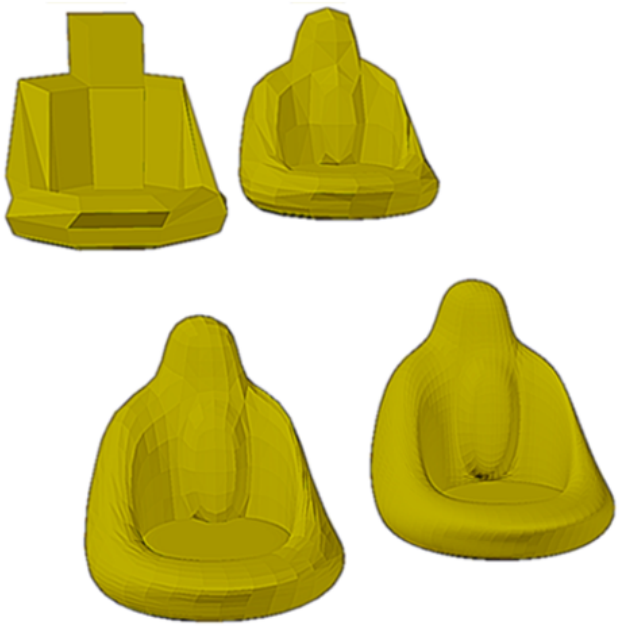
Increase the roundness of mesh objects by increasing the smoothness levels.

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.

NOTE 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 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.

To decrease the smoothness of a mesh object

1. Click Mesh 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.

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.

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.

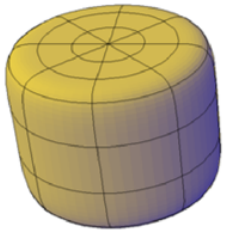
Refine Mesh Objects or Subobjects

Refine a mesh object or subobject to convert underlying facets to editable faces.

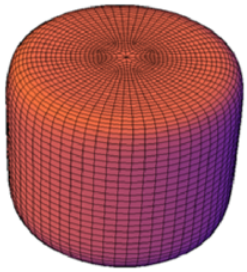
You can refine any mesh that has a level of smoothness of 1 or higher.

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.



smoothness level = 3
before refinement

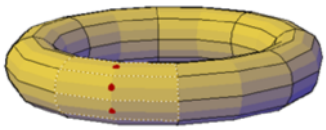


new smoothness level = 0
after refinement

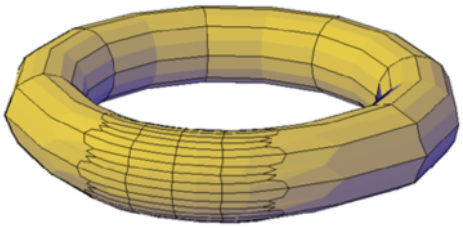
In addition to increasing the number of faces, refining a mesh object resets its level of smoothness back to the baseline. Therefore, an object might appear to be smoothed, but its smoothness level can still equal 0 (zero).

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.



face before refinement

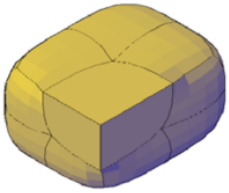


face after refinement

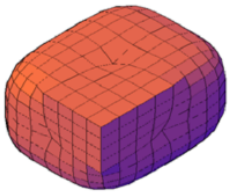
Refining a mesh face does not affect the overall smoothing level of the mesh object. Unlike a refined mesh object, refined faces can be refined again immediately. With mesh face refinement, you can target smaller areas for detailed modeling.

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.



smoothness level = 2
crease value = 4



mesh after refinement
smoothness level = 0
crease value = 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 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.

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 tab ➤ Mesh panel ➤ Refine Mesh. 
3. 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.

Quick Reference

Commands

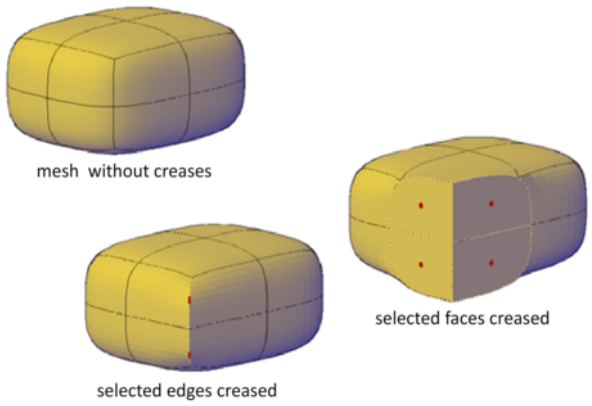
MESHREFINE

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 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).

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.

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 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.

Quick Reference

Commands

MESHCREASE

Sharpens the edges of selected mesh subobjects.

MESHUNCREASE

Removes the crease from selected mesh faces, edges, or vertices.

PROPERTIES

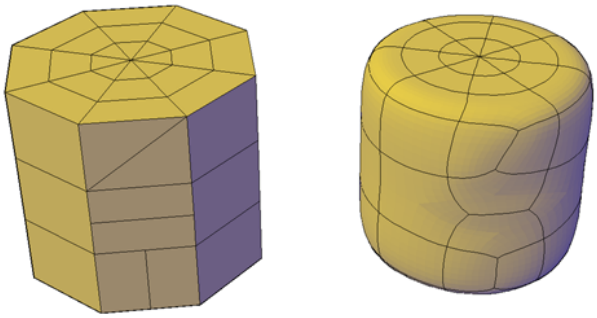
Controls properties of existing objects.

Modify Mesh Faces

Split, extrude, merge, collapse, or spin mesh faces to modify their shapes.

Split a Mesh Face

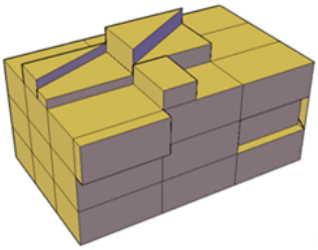
You can split a mesh face to make custom subdivisions. Use this method to prevent deforming a larger area for small modifications.



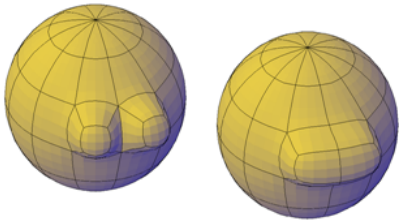
Because you specify the start point and end point of the split, this method also gives you control over the shape of the two new faces. Use the Vertex option to snap automatically to the vertices of the face. If you plan to split a face to create—and then spin the edge of—two triangular faces (MESHSPIN), use the Vertex option to ensure precision.

Extrude Mesh Faces

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. However, when you extrude mesh faces, the MESHEXTRUDE command provides an option that sets whether adjacent faces are extruded individually or whether their shared edges remained joined.



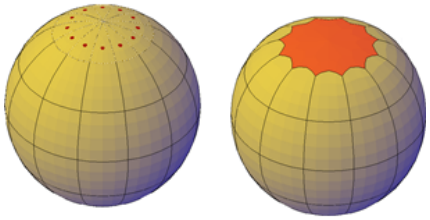
You cannot create joined extrusions for mesh faces in which only the vertices are shared.

For more information about extrusion, see Create a Solid or Surface by Extruding on page 1034.

Reconfigure Adjacent Mesh Faces

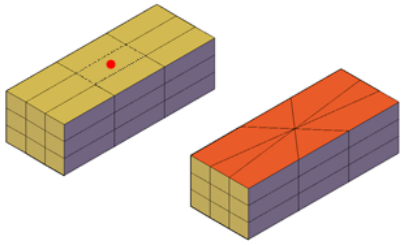
You can extend your editing options by reconfiguring adjacent faces. Several options are available:

- **Merge adjacent faces.** Combine adjacent faces to form a single face. Merging works best with faces that are on the same plane.

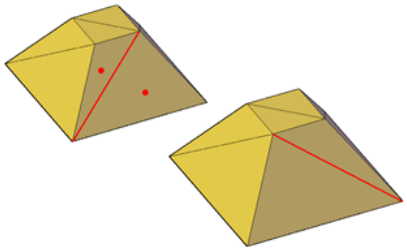


  Although you can merge faces that wrap a corner, additional modifications to the resulting mesh object can have unexpected results.

- **Collapse the mesh vertices.** Merge adjacent vertices of surrounding faces form a single point. The selected face is removed.



- **Spin edges of triangular faces.** Rotate an edge that is shared by two triangular faces. The shared edge spins to extend from the opposite vertices. This activity works best when the adjoined triangles form a rectangular, not a triangular, shape.



See also:

- Tips for Working with Mesh on page 1261
- Create a Solid or Surface by Extruding on page 1034

To split a mesh face

1. Click Mesh tab ➤ Mesh Edit panel ➤ Split Face.
2. Click the mesh face that you want to split.
3. To snap to a vertex of the face, enter **v** (Vertex).
4. Click the location where you want to start the split.
5. Click a second location to end the split.

   The mesh face is split along the boundary you specified. Surrounding faces are adjusted.

To extrude mesh faces

1. Click Mesh tab ➤ Mesh Edit panel ➤ Extrude Face.
2. To review or change the extrusion method, press **s** (Setting) and set one of the following options:
   - To keep adjacent faces joined as you extend them, enter **y** (Yes).
   - To extend adjacent faces separately, enter **n** (No).

3 Click one or more mesh faces and press Enter.

4 Specify the amount of extrusion using one of the following methods:

- (Single face extrusion only) Press Enter to set the extrusion dynamically.
- Enter a value to indicate the height or depth of the extrusion and press Enter.

The selected face is extruded.

To merge mesh faces

1 Click Mesh tab ➤ Mesh Edit panel ➤ Merge Face. 

2 On a mesh object, select two or more mesh faces and press Enter.

The selected faces are combined to form a single face.

To collapse mesh faces

1 Click Mesh tab ➤ Mesh Edit panel ➤ Mesh Edit drop-down ➤ Collapse Face or Edge. 

2 Select a mesh face or edge.

NOTE If you cannot select a face or edge, verify whether subobject selection filtering is turned on for a different subobject type. (Right-click the drawing area and click Subobject Selection Filter.)

The vertices of the face or edge that you selected are collapsed to a single point.

To spin the shared edge of triangular mesh faces

1 Click Mesh Modeling tab ➤ Mesh Edit panel ➤ Mesh Edit drop-down ➤ Spin Triangle Face. 

2 Select two triangular mesh faces that are adjacent to one another.

The edge shared by the original triangles rotates, or spins, to connect two different vertices.

Quick Reference

Commands

MESHCOLLAPSE

Merges the vertices of selected mesh faces or edges.

MESHEXTRUDE

Extends a mesh face into 3D space.

MESHMERGE

Merges adjacent faces into a single face.

MESHSMOOTHLESS

Decreases the level of smoothness for mesh objects by one level.

MESHSMOOTHMORE

Increases the level of smoothness for mesh objects by one level.

MESHSPIN

Spins the adjoining edge of two triangular mesh faces.

MESHSPLIT

Splits a mesh face into two faces.

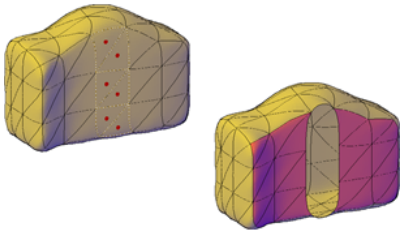
Create and Close Mesh Gaps

Delete mesh faces or close gaps in mesh objects.

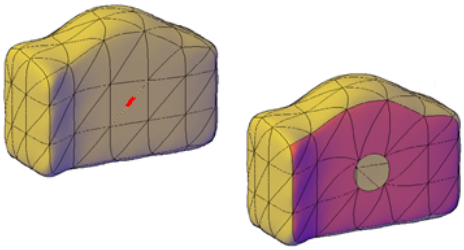
Remove Mesh Faces

You can press Delete or use the ERASE command to remove mesh faces. The removal leaves a gap in the mesh.

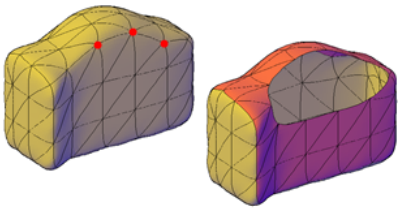
- Deleting a face removes only the face.



- Deleting an edge removes each adjacent face.



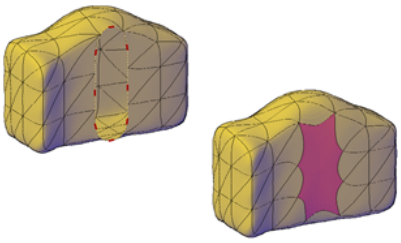
- Deleting a vertex removes all faces that are shared by the vertex.



If removal of a mesh face creates a gap, the mesh object is not "watertight." It can be converted to a surface object, but not to a 3D solid object.

Close Gaps in Mesh Objects

If a mesh object is not watertight due to gaps, or holes, in the mesh, you can make it watertight by closing the holes. The cap, or new face, spans the boundary formed by the mesh edges that you specify (MESHCAP).



This process works best when all edges are on the same plane. The edges you select as boundaries cannot be shared by two faces. For example, you cannot close the center hole in a mesh torus.

NOTE You can sometimes close gaps in mesh by smoothing the object, by using MESHCOLLAPSE, or by splitting adjacent faces (MESHSPLIT).

See also:

- Tips for Working with Mesh on page 1261

To delete faces from a mesh object

1. Press Ctrl+click one of the following mesh subobject types:
   - To remove only that face, click the face.
   - To remove adjacent faces, click their shared edge.
   - To remove all faces that share a vertex, click the vertex.

   NOTE If you cannot select the subobject you want, verify whether subobject selection filtering is turned on for a different subobject type. (Right-click in the drawing area and click Subobject Selection Filter.)

2. Press Delete.

NOTE You can also remove mesh faces with the ERASE command.

To repair a hole in a mesh object

1. Click Mesh tab ➤ Mesh Edit panel ➤ Close Hole. 

2 Select contiguous edges to serve as the boundary for the mesh face that will span the gap.

Quick Reference

Commands

ERASE

Removes objects from a drawing.

MESHCAP

Creates a mesh face that connects open edges.

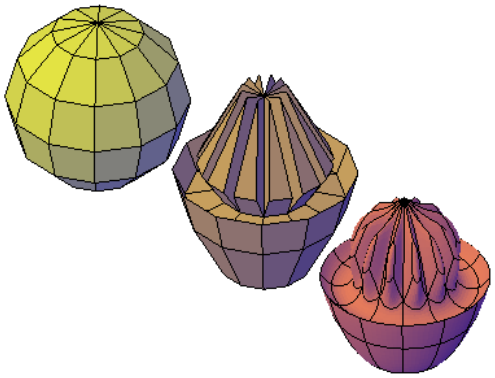
Tips for Working with Mesh

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.)

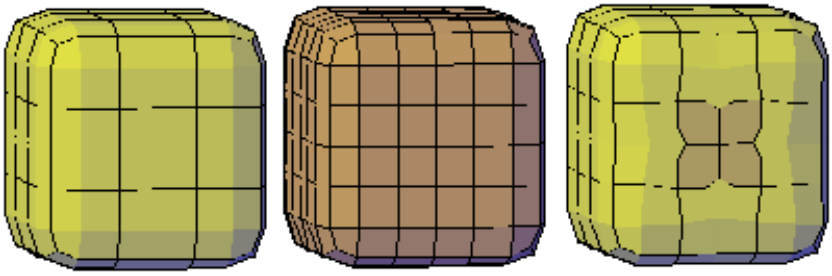


mesh sphere modeled by grip editing and extrusion, then smoothed

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.

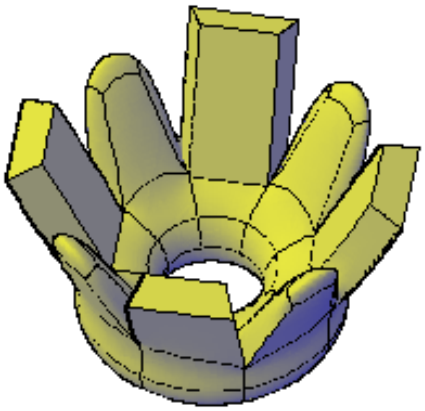


mesh box, refined mesh box, and mesh box with one face refined

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.

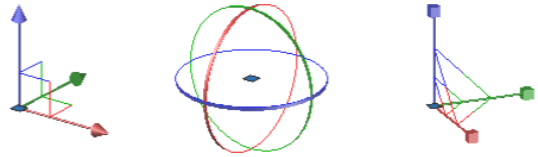


extruded faces on mesh torus, creased and not creased

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

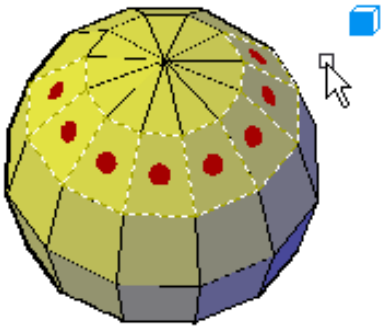
For example, you can rotate and scale an individual face using the 3D Move, Rotate, and Scale gizmos.

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 for selection.



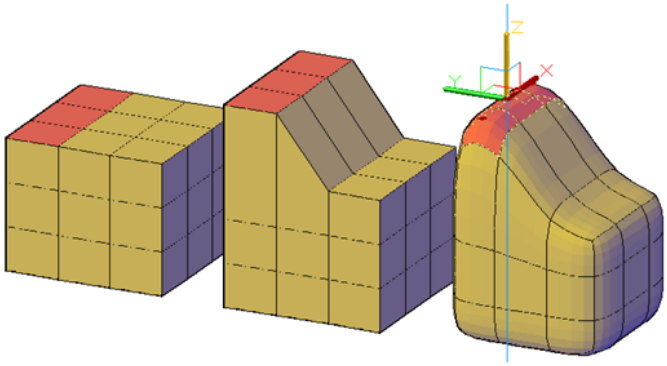
mesh faces selected when the face subobject selection filter is on

A filter is especially valuable for selecting mesh vertices, which are not highlighted as you move the mouse over them.

In order to select the entire mesh object, you need to turn off the subselection filters.

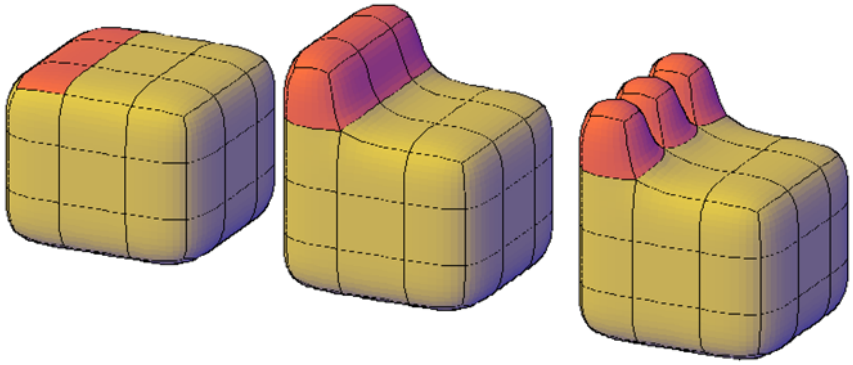
Model by extruding faces.

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

Mesh extrusion, however, inserts additional faces to close the gap between the extruded face and its original surface. With mesh extrusion, you can set whether adjacent faces are extruded as a unit (joined) or separately (unjoined).



mesh faces extruded, then smoothed

If you are working on an object that has not been smoothed, try smoothing it periodically to see how the extrusion 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.

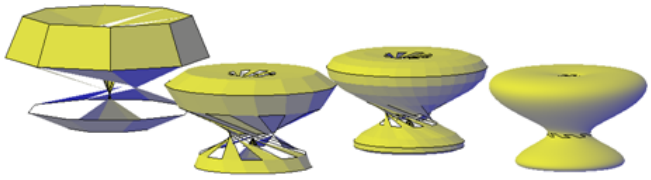
Keep in mind that not all conversions retain complete fidelity to the shape of the original object. Avoid switching between object types more than once, if possible. If you notice that the conversion modifies the shape of the object in an unacceptable way, undo the conversion and try again with different 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 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.

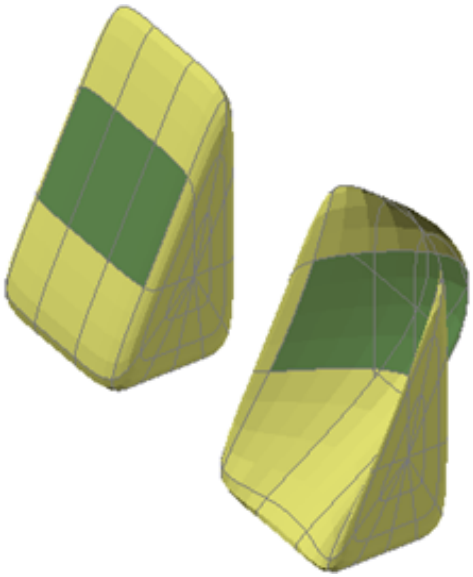


mesh torus that has been twisted using 3D Rotate at various smoothing levels

  You can also close holes by using MESHCAP.

  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.

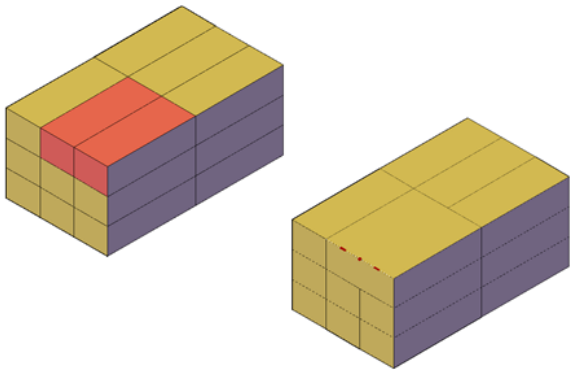


mesh wedge with front faces dragged past the back faces

Mesh objects that cannot be converted to solids can often be converted to surfaces instead.

Avoid merging faces that wrap a corner

When you merge faces, you can create a mesh configuration in which the merged face wraps a corner. If a resulting face has a vertex that has two edges and two faces, you cannot convert the mesh to a smooth 3D solid object.



One way to resolve this problem is to convert the mesh to a faceted solid instead of a smooth solid. You might also be able to repair the problem by splitting the adjacent faces, starting at the shared vertex (MESHSPLIT).

Modify 3D Solids and Surfaces Using Inventor Fusion

Open and modify 3D solids and surfaces directly in Autodesk Inventor Fusion, a 3D direct manipulation modeler.

Inventor® Fusion provides direct modeling capabilities for rapid design changes without limits.

Without exiting AutoCAD, you can launch Inventor Fusion, use it to modify 3D AutoCAD solid and surface objects, and then continue working on the modified objects in AutoCAD.

To modify 3D AutoCAD objects using Fusion

1. Select one or more 3D solid or surface objects.
2. Click Plug-Ins tab ➤ Inventor Fusion panel ➤ Edit In Fusion.
3. Make the required changes in Fusion.
4. In Fusion, click Return To AutoCAD.

Quick Reference

Commands

EDITINFUSION

Edits solids and surfaces in Inventor Fusion.

Create Sections and Drawings from 3D Models

26

Create cross sections, cutting planes, flattened views, and 2D drawings of 3D objects.

Create Drawings from 3D Models

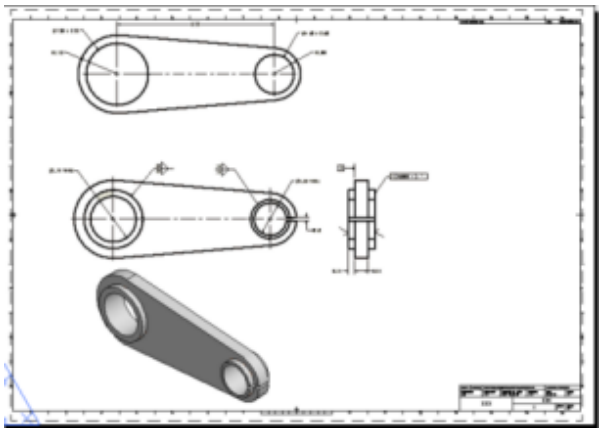
Create drawings from a variety of 3D models.

Create Drawings from AutoCAD and Inventor 3D Models

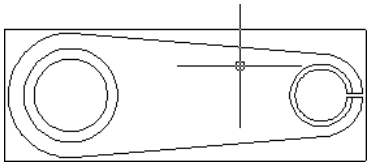
Create associative drawings in layouts from AutoCAD and Autodesk Inventor 3D models.

Workflow for Creating Associative Drawings from 3D Models

You can generate drawings from 3D models, which are associative to the model they were generated from. The model can be a model space 3D solid or surface, or an Autodesk Inventor 3D model.



The basic building block of a drawing is a drawing view.

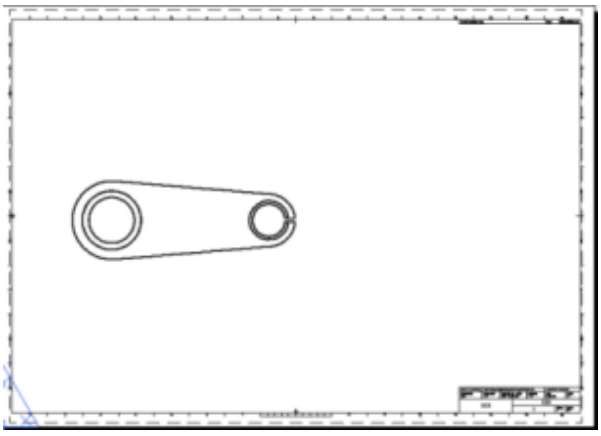


A drawing view is a rectangular object that contains a 2D projection of the 3D model. The VIEWBASE and VIEWPROJ commands generate drawing views from the 3D model.

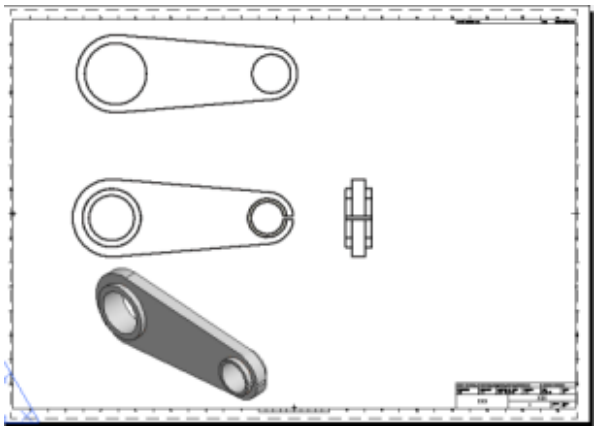
The border of the drawing view is visible only when you create the view or when you move the cursor over it. Although the view border resides on the current layer, it does not print.

The view geometry is always visible, and is created on a pre-defined set of layers (See Change Layer Properties of View Geometry on page 1292 for details). Since the view geometry is associative with the source 3D model, you cannot select view geometry to modify them.

The first view you place on a drawing is called a base view. Base views are drawing views that are derived directly from the 3D model.

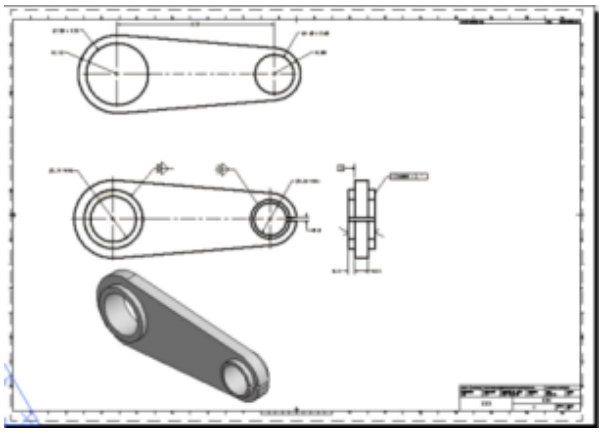


Once a base view is placed in a layout, you generate projected views from it. Unlike a base view, projected views are not directly derived off the 3D model. Instead, they are derived from a base view (or another projected view that already exists in a layout). The projected views maintain a parent-child relationship with the view it was generated from. Most settings of a child view are derived from the parent.

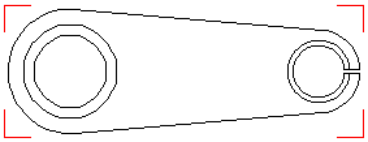


If necessary, you can create more than one base view in a layout. This feature gives you the ability to create views of multiple parts or assemblies in the same drawing.

Once you create the views, you can add notes, dimensions, symbols, centerlines, and other annotations as required.



When the source 3D models change, drawing views derived from that model become out-of date. AutoCAD highlights out-of-date drawing views by displaying red markers on the corners of the view border.



To synchronize an out-of-date view with its source 3d model, update the view.

You can use the EXPORTLAYOUT command to export a layout containing drawing views to the model space of a new drawing. The exported drawing is no longer associative with the 3D model, and hence you can edit view geometry without any restrictions.

Quick Reference

Commands

EXPORTLAYOUT

Creates a visual representation of the current layout in the model space of a new drawing.

VIEWBASE

Creates a base view from model space or Autodesk Inventor models.

VIEWPROJ

Creates one or more projected views from an existing drawing view.

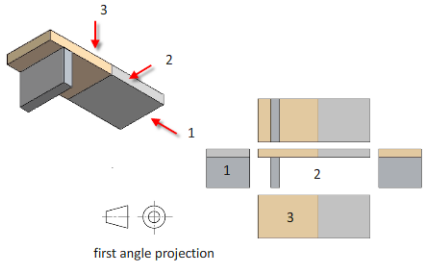
Set Defaults for Drawing Views from 3D Models

Customize the default settings for drawing views.

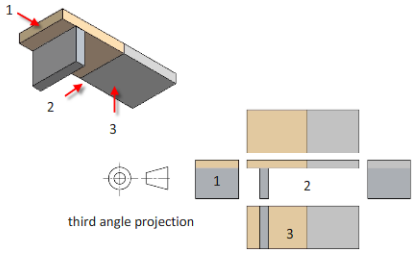
To ensure that drawing views follow the correct drafting standards, you customize the default settings using the Drafting Standards dialog box. The settings you specify impact all new drawing views you create. They have no effect on any existing drawing view.

The Drafting Standards dialog box displays when you run the VIEWSTD command. Using this dialog box you can control:

- The projection type
  The ISO standard uses first angle projection. The ANSI standard uses third angle projection. You select the projection angle corresponding to the drafting standard the drawing must comply with.



first angle projection



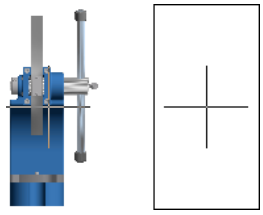
third angle projection

- The thread style
  The ISO standard uses a partial circle to indicate a thread edge. The ANSI standard uses a full circle to indicate a thread edge. You select the thread

angle corresponding to the drafting standard the drawing must comply with.



- The preview type
  The preview is a temporary graphic that is displayed with the cursor at the time you place a view in the drawing area. Shaded previews provide visual feedback to enable you to select the desired model orientation during view creation. However, for large models shaded previews can take time to generate. For such cases you can change the temporary graphic to a bounding box, instead of using a shaded preview.



  Even though this setting is not related to a drafting standard, it is a setting you must specify before you create a drawing view.

  You can also specify the resolution of the shaded preview. While higher resolutions provide better quality previews, resolutions above 150 dpi may not be achievable for larger models. When the specified resolution is not achievable, the program automatically drops the quality to an achievable level.

To specify the default projection angle

1. Click Annotate tab ➤ Drawing Views panel ➤ Dialog Box Launcher.
2. In the Drafting Standards dialog box, in the Projection type section, click the desired projection angle.
3. Click OK.

To specify the default thread style

1. Click Annotate tab ➤ Drawing Views panel ➤ Dialog Box Launcher.

2 In the Drafting Standards dialog box, in the Thread style section, click the desired thread style.

3 Click OK.

To specify the preview style

1 Click Annotate tab ➤ Drawing Views panel ➤ Dialog Box Launcher.

2 In the Drafting Standards dialog box, from the Preview type list, select the desired preview style.

3 If you selected Shaded, from the Shaded view quality list, select the desired resolution.

NOTE Resolutions above 150 DPI may not be achievable for larger models. If you use a high resolution setting, at view creation time, the resolution adjusts automatically for larger models.

4 Click OK.

Quick Reference

Commands

VIEWSTD

Defines the default settings for drawing views.

Create Base Views from 3D Models

Workflow for creating base views for AutoCAD 3D models and Inventor models.

The first drawing view you place in a layout is a base view. Base views define the scale, display settings, and alignment for subsequent views.

The base view creation process begins when you click the Base View button on the Ribbon, from a layout. The process of selecting the source 3D model differs depending on whether the source is an AutoCAD 3D model or an Inventor model. Once the 3D model to use is determined, the following takes place:

- The Drawing View Creation contextual ribbon tab is displayed (if the ribbon is on).

- A scaled preview of the model appears on the cross hairs.
- The command line prompts you to place the base view in model space.

All the options that are available on the Drawing View Creation contextual ribbon tab are available on the command line as well. The purpose of the command line is to assist you when the ribbon is turned off. However, if the ribbon is turned on, we recommend that you use the ribbon. This because the ribbon hides options or makes them unavailable depending on its relevancy to the selected model. The command line, on the other hand, displays all options, even the options that do not apply to the selected model.

The command does not exit as soon as you place the base view in the drawing area. You can continue to change option settings until you click the OK button on the ribbon. A typical use of this capability is described below.

You place the base view in the drawing and you realize that the view is too large. You then use the Scale option on the ribbon and change the scale. Once you change the scale, you find that the view is not positioned correctly. You then use the Move command on the ribbon to move the view. Once you are done you click OK on the ribbon.

By default, the program prompts you to create projected views soon after you are done with the base view. However, if the model you are creating is large, creating base views and projected views in the same command session can take time. In such a scenario, you can use the Type command line option and specify that you are creating only a base view.

Create Base Views from AutoCAD 3D Models

Additional information specific to creating base views from AutoCAD 3D models.

The Base View button on the Ribbon (VIEWBASE command), creates base views from model space solids and surfaces.

The base view contains all visible solids and surfaces in model space. It does not contain any solid or surface that exists on a layer that is frozen or turned off. Similarly, if a solid or surface is isolated, only the isolated solid or surface is included the view.

If frozen layers are thawed, turned off layers are turned on or the isolation is ended, the base view does not update automatically. You must explicitly use the Update View (VIEWUPDATE) command to bring in the changes.

To create a base view from model space solids and surfaces

1 Click Annotate tab ➤ Drawing Views panel ➤ Base View 

NOTE

- You can use this command only from a layout tab.
- If the layout tab contains a viewport, we recommend that you delete it before you invoke this command.

2 In the Orientation panel of the Drawing View Creation contextual ribbon tab, select the orientation for the base view.

3 In the Appearance panel of the Drawing View Creation contextual ribbon tab, specify the scale and view style.

4 Click in the drawing area to indicate the location to place the base view.

5 Click View Creation tab ➤ Create panel ➤ OK.

Quick Reference

Commands

VIEWBASE

Creates a base view from model space or Autodesk Inventor models.

Create Base Views from Inventor Models

Additional information specific to creating base views from Autodesk Inventor models.

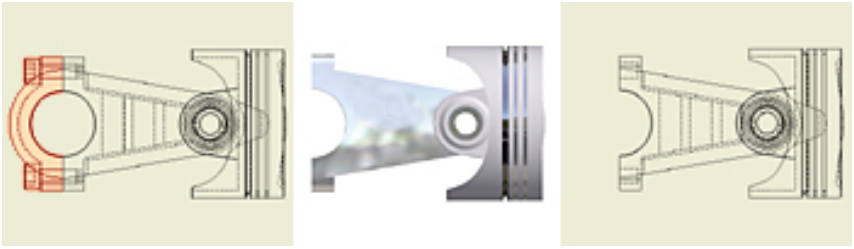
The base view creation process begins when you click the Base View button on the Ribbon (VIEWBASE command), from a layout. If model space does not contain any solids or surfaces, the command displays a file open dialog box to enable you to select an Inventor model.

In addition to selecting the Inventor model, the dialog box provides access to some common representation types used by assemblies. You can choose to ignore them at this time and specify them later. However, specifying these

options during file open improves performance of large assemblies. This is because the command can load only the data necessary for your selection.

Like AutoCAD named views, Inventor models contain named *view representations*. However, unlike AutoCAD, Inventor models have several types of view representations, which you can mix and match. While creating base views, you must specify which of these view representations you are creating the base view for. The view representation types are:

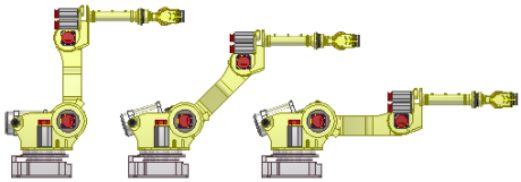
Design view representations



A design view representation is a display configuration that you can recall by name. Design views control component visibility and material overrides, usually to present an uncluttered view of the components you intend to show in that view.

When you create a base view, you must specify which design view representation you are creating it for.

Positional representations



A positional representation is one snapshot in a collection of snapshots that show an assembly in various positions. You can access each positional representation by name.

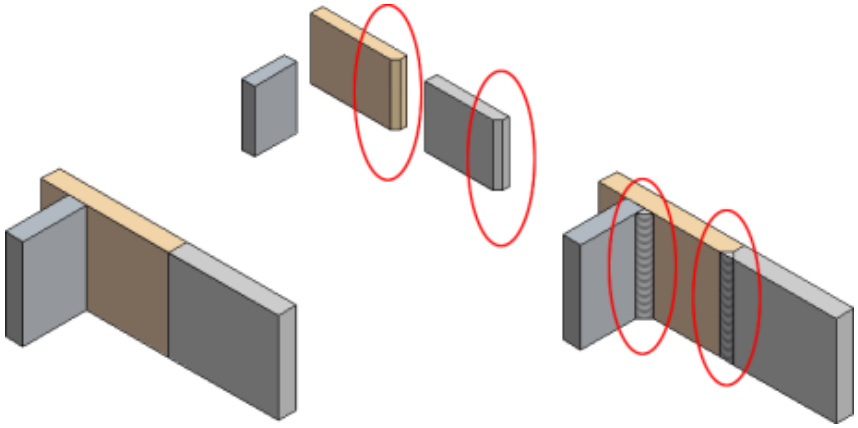
When you create a base view, you must specify which design view representation you are creating it for.

Level of detail

A named representation saved in an Inventor assembly file, where unneeded components are suppressed or replaced with a single part representation in order to simplify the model.

When you create a base view, you must specify which level of detail representation you are creating it from.

Weldment states



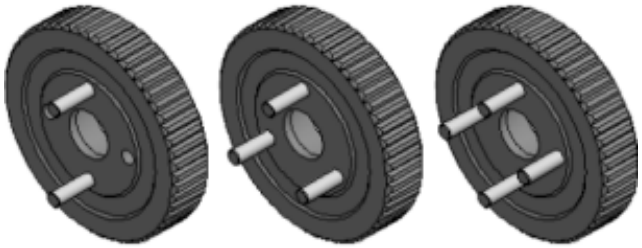
A weldment is an inseparable assembly formed by welding together two or more pieces. Inventor models can show the weldment in different states. The "Assembly" state displays the weldment in its unwelded state. The "Welds" state displays the weldment after the pieces have been welded in place. The "Machining" state shows the weldment after post welding machining operations (if any). Additionally, you can display each individual piece with any weld preparation it may have.

The illustration shown above depicts a weldment in Assembly state, each piece in preparatory state and the weldment in Welds state.

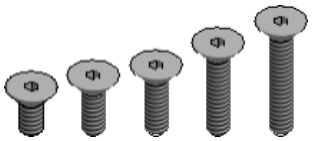
When you create a base view, you must specify which state you are creating it from.

Member

The member view representation type applies only to iAssembly factories or iPart factories.



An iAssembly factory is family of assemblies, derived off a single model, by varying a set of unique identifiers. For example, a family of flywheels with varying number of shoes.



An iPart factory is a family of parts, derived off a single model by varying parameters such as size or material. For example, a family of screws of varying lengths and diameter.

When creating a base view, you must specify which member of the family you are creating the base view for.

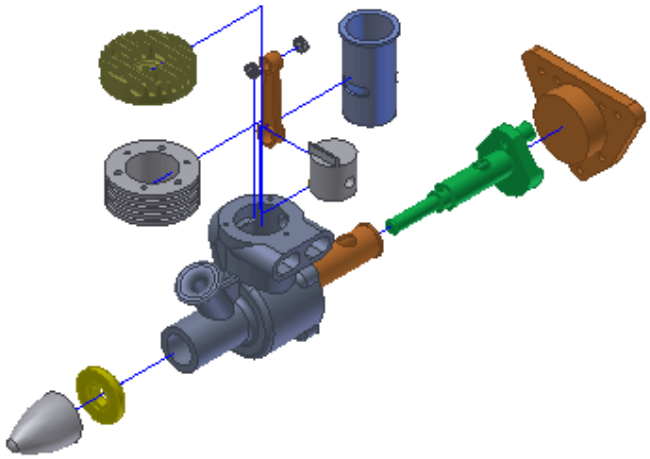
Sheet metal



Sheet metal views are available only on sheet metal part files. When creating a base view of a sheet metal part, you specify if it is to be created from the folded model or the flat (unfolded) pattern.

NOTE Flat patterns are available only if they are defined in the sheet metal part file.

Presentation view



A presentation view is an exploded view of an assembly, created to demonstrate how parts in an assembly interact and fit together.

When you create a base view, you must specify which presentation view you are creating it for.

NOTE Presentation views are available only if the model you are using is a presentation (*.ipn) file.

To create a base view from an Inventor model

1. Verify that model space is empty.
2. Click the layout tab you intend to create the drawing in.
3. Click Annotate tab ➤ Drawing Views panel ➤ Base View 

   NOTE

   - You can use this command only from a layout tab.
   - If the layout tab contains a viewport, we recommend that you delete it before you invoke this command.

4 In the Select File dialog box, select the model to generate the base view from and click Open.

5 In the Representation panel, select the view representations you want to show in the base view.

NOTE Only the view representations used by the model you selected are available on the ribbon. For example, if you selected an assembly, the Weldment, Member, Sheet Metal and Presentation View representations are disabled.

6 In the Orientation panel of the Drawing View Creation contextual ribbon tab, select the orientation for the base view.

7 In the Appearance panel of the Drawing View Creation contextual ribbon tab, specify the scale and view style.

8 Click in the drawing area to indicate the location to place the base view.

9 Click View Creation tab ➤ Create panel ➤ OK.

Quick Reference

Commands

VIEWBASE

Creates a base view from model space or Autodesk Inventor models.

Create Projected Views from Existing Drawing Views

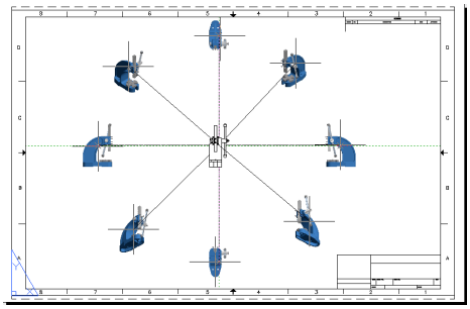
Workflow for creating projected views from existing views.

A projected view is an orthographic or isometric projection of an existing base view or projected view.

The VIEWPROJ command prompts you to select an existing view as a parent view. Any existing view can be selected, as long as it is not out-of-date or unresolved on page 1294. The projected view inherits all its properties from the parent, including the projection angle (first angle or third angle). You can change some of the properties after you create the projected view.

Once you select the parent view, a preview of the projected view is attached to the cursor. The orientation of a projected view depends on its location in

relation to the parent view. The preview updates dynamically, to reflect the orientation, as you move the cursor.



After you place a projected view in the drawing area, the command prompts repeat. Hence, you can place multiple projected views within the same command session. You exit the command by selecting the Exit command line option.

To create projected views



1. Click Annotate tab ➤ Drawing Views panel ➤ Projected View.
2. Click the view you want to use as the parent view.
3. Move the preview to the desired location and click to place the view.

   NOTE As you move the preview the orientation of the projected view changes to reflect its relationship to the parent view.

4. Repeat step 4 until all the required projected view are created.
5. Press ENTER.

Quick Reference

Commands

VIEWPROJ

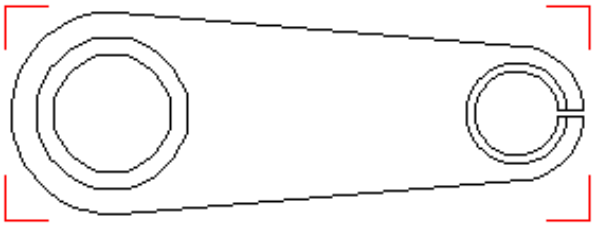
Creates one or more projected views from an existing drawing view.

Update Drawing Views when Source 3D Models Change

Workflow to make out-of-date drawings current.

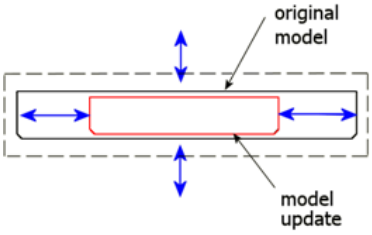
While you work on a drawing containing drawing views, the program periodically checks whether the referenced 3D source model has changed. If the source has changed, a balloon message displays next to the drawing views status bar icon in the lower right of the application window. Click the link in the balloon message to update all out-of-date views.

If you close the balloon without updating the views, a warning badge is attached to the drawing view status bar icon. Furthermore, AutoCAD highlights out-of-date drawing views by placing red markers on the corners of the view border. You update an out-of-date view by right-clicking it and selecting Update from the menu. Alternatively, you can use the Update View command (or use the Update All Views command to update all out-of-date views) from the drawing views ribbon panel.



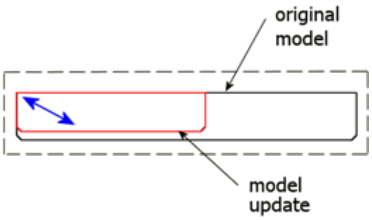
Unlike Xrefs, drawing views do not update automatically when you open a drawing. You must explicitly invoke the AMVIEWUPDATE command to update drawing views.

Depending on the change in the source model, the view may grow or shrink. There are two ways by which you can anchor a view in the drawing.



Centered

You can choose to center justify the view, so that the view grows or shrinks about its center.



Fixed

You can also choose the “fixed” option, which forces unchanged model geometry to remain in the location it was prior to the update operation. Often, this is preferred as less shift occurs.

If you update a view after you have added dimensions to the view geometry, dimensions are likely to become disassociated. You are then prompted to step through the affected dimensions and attach the extension-line origin points of each disassociated dimension to the corresponding geometry. If necessary, you can skip reassociating dimensions at this time and use the DIMREASSOCIATE command later.

To update an out-of-date view

1. Click Annotate tab ➤ Drawing Views panel ➤ Update View drop-down ➤ Update View. 
2. Click the view you want to update.
3. Press ENTER.

To specify how to anchor a view

1. Double-click the view you want to anchor.
2. On the Drawing View Editor contextual ribbon tab, click View Options.
3. From the View Justification list, select the desired anchor option.
   - To anchor the view so that it grows or shrinks around its center, select Centered.
   - To anchor the view such that unchanged model geometry does not move, select Fixed.
4. On the Drawing View Editor contextual ribbon tab, click OK.

Quick Reference

Commands

DIMREASSOCIATE

Associates or reassociates selected dimensions to objects or points on objects.

VIEWUPDATE

Updates drawing views that have become out-of-date because the source model has changed.

Edit Existing Drawing Views

Perform a variety of edit operations on drawing views.

Edit Drawing View Properties

Edit properties such as scale, view options, and view representation, which control the appearance of drawing views.

The editing process begins when you double-click a drawing view or when you enter the VIEWEDIT command at the command prompt. Regardless of the view's typeor state, the view is selected and the Drawing View Editor contextual ribbon tab is displayed. If the ribbon is turned off, use command line options to change the properties.

NOTE Although you can inspect properties of a drawing view from the properties palette and quick properties palette, you cannot change most of them.

If the view is out-of-date it automatically updates when you edit it.

If the selected view is a base view, most properties that were available within the ribbon at creation time can be changed.

If the selected view is a parent view, when it is edited, the changes are applied to child views as well. However changes to design view representations are not applied to child views.

If the selected view is a child view, only the scale and style properties are available. The default values for these properties are set to "From parent". If you override these properties, they cease to follow changes made to the parent view.

In general, the options in the Representation Panel of the Drawing View Editor contextual ribbon tab are available only for base views. The Appearance panel is available for both base views and projected views.

When a model is large or complex, applying a property can take time. If you turn Defer Updates, property changes are not applied immediately. Instead, they are applied only when you click OK, or turn off Defer Updates. This saves you the trouble of having to wait for the display to update, each time you change a property.

To change a property of a drawing view

1. Double-click the view you want to edit.
2. On the Drawing View Editor contextual ribbon tab, in the list corresponding to the property you want to change, select the new property value.
3. On the Drawing View Editor contextual ribbon tab, click OK.

To turn Defer Updates on or off

1. Double-click the view you want to edit.
2. On the Drawing View Editor contextual ribbon tab, in the Edit panel, check the status of the Defer Updates button.

   If the button is pressed, Defer Updates is on. If the button is released, Defer Updates is off.
3. Click Defer Updates to change its status.
4. On the Drawing View Editor contextual ribbon tab, click OK.

   The state of the Defer Updates button is persistent. The button remembers its status until you change it again.

Quick Reference

Commands

VIEWEDIT

Edits an existing drawing view.

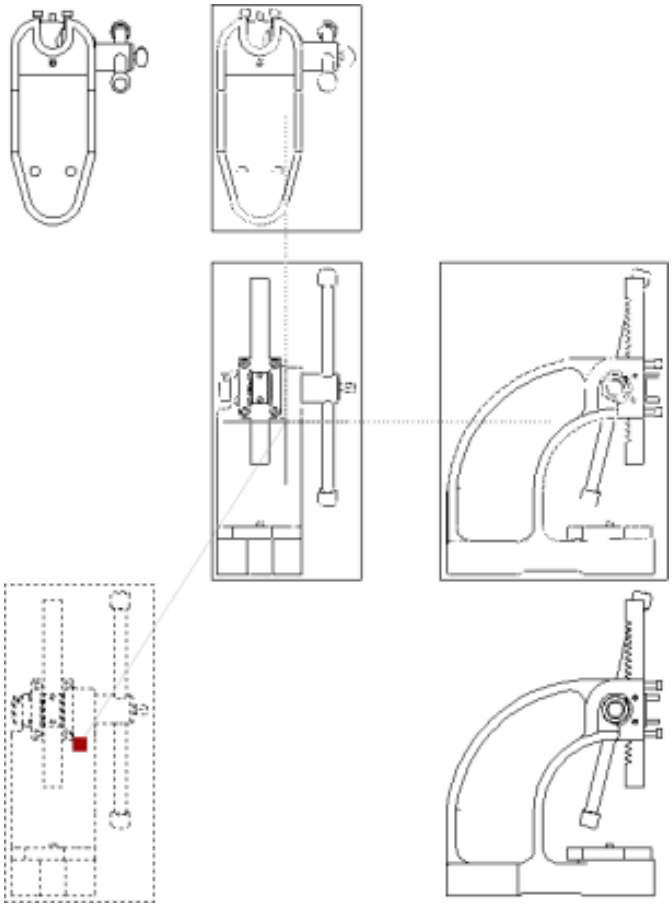
Move, Rotate, or Erase drawing views

Move drawing views to a different location on a layout, rotate the view geometry, or delete drawing views.

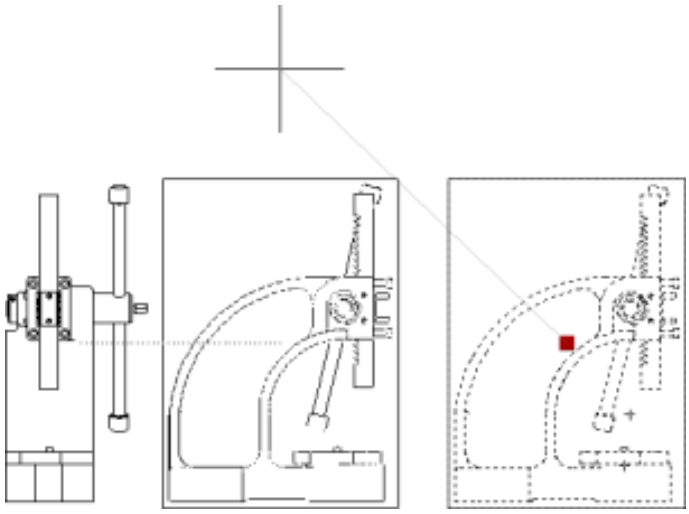
Move Drawing Views

You can use the MOVE command to move drawing views.

Orthogonal views are constrained to be aligned to their parent views, while isometric views are not. Consequently, if you move a parent view, all orthographic child views also move with it to maintain alignment.



If you move a child view (orthographic), it is constrained to maintain alignment with the parent.

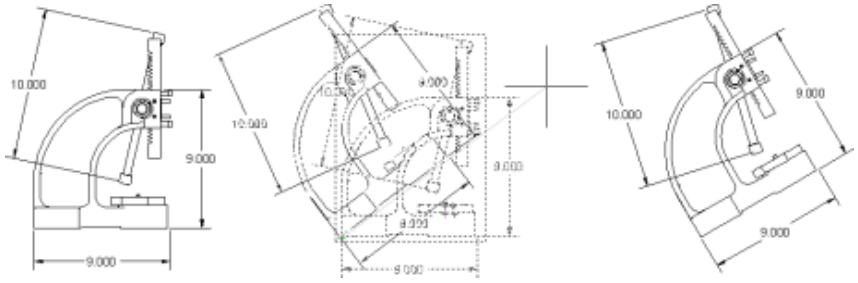


If required, you can display the grip menu of a view and select Break Alignment to remove the alignment constraint. A child view that you moved in this manner can be realigned by using the Repair Alignment option in the grip menu of the view.

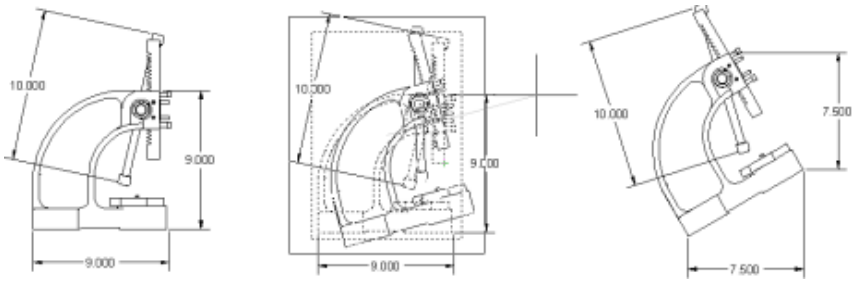
Rotate Drawing Views

You can use the ROTATE command to rotate drawing views.

If you select dimensions as part of the selection set, they too are rotated.



When you do not include dimensions within the selection set, only aligned dimensions rotate with the view. Aligned dimensions continue to remain aligned while horizontal, vertical, and rotated dimensions continue to remain as they are.



Rotating a view breaks all alignment constraints it has with other views.

Erase Drawing Views

You can erase a drawing view using the ERASE command.

If you erase a parent view, its child views are not erased. The child views automatically assume the role of a parent view. Although child views become parent views, they do not become base views. Therefore, the "View type" property within the Properties palette continues to display the "Projected" value. Furthermore the options on the Drawing View Editor contextual ribbon tab that apply to base views, continue to be unavailable for child views.

When annotations are associated with the view, unless you explicitly select them, they are not erased.

Impact of Copy Command and Layout Command

Drawing view objects are filtered out from the COPY command. Hence you cannot use the COPY command on drawing views.

A layout containing drawing views can be copied using the LAYOUT command's COPY option. When copying a layout containing views, all views are filtered from the layout; the new layout is copied without the views.

To move a drawing view

1. Click the view you want to move. A grip appears at the center of the view.
2. Move the cursor over the grip. A menu is displayed.
3. Click Stretch.
4. Move the preview to the desired location and click to place the view.

To rotate a drawing view

1. Click the view you want to rotate. A grip appears at the center of the view.
2. Move the cursor over the grip. A menu is displayed.
3. Click Rotate.
4. Move the preview to the desired location and click to place the view, or specify the rotation angle and press ENTER.

To remove the alignment constraint on a child view

1. Click the view you want to remove the constraint from. A grip appears at the center of the view.
2. Move the cursor over the grip. A menu is displayed.
3. Click Break Alignment. The view can now be moved anywhere on the layout.

To reinstate the alignment constraint on a child view

1. Click the view you want to reinstate the constraint on. A grip appears at the center of the view.
2. Move the cursor over the grip. A menu is displayed.
3. Click Repair Alignment. The view moves back into alignment with its parent.

Quick Reference

Commands

ERASE

Removes objects from a drawing.

LAYOUT

Creates and modifies drawing layouts.

MOVE

Moves objects a specified distance in a specified direction.

ROTATE

Updates drawing views that have become out-of-date because the source model has changed.

Change Layer Properties of View Geometry

Change the appearance of view geometry and comply with company layer naming standards.

When you create a drawing view, the program draws view geometry on pre determined layers, depending on what the geometry represents. For example, a groove on the front side of a part is drawn on a layer named “visible” in the front view. The same groove is drawn on a layer named “hidden” in the back view, because it represents an invisible edge.

The following table shows the layers on which each entity is drawn on..

Entity Represented by Geometry	Layer drawn on	Model Source	
		Inventor	Model Space
Interference Edge (Hidden)	Hidden Narrow	Yes	Yes
Interference Edge (Visible)	Visible Narrow	Yes	Yes
Presentation View Trail Line	Tweak Trail	Yes	No
Reference Part Edge	Reference	Yes	No
Sheet Metal Bend Centerline Down	Centerline	Yes	No
Sheet Metal Bend Centerline Up	Centerline	Yes	No
Sheet Metal Bend Extent	Bend Extent	Yes	No
Sheet Metal Roll Centerline	Centerline	Yes	No
Simplified Representation (Hidden)	Hidden Narrow	Yes	No
Simplified Representation (Visible)	Visible Narrow	Yes	No

Entity Represented by Geometry	Layer drawn on	Model Source	
		Inventor	Model Space
Tangent Edge (Hidden)	Hidden Narrow	Yes	Yes
Tangent Edge (Visible)	Visible Narrow	Yes	Yes
Thread Ends (Visible)	Visible	Yes	No
Thread Ends (Hidden)	Hidden	Yes	No
Thread Lines (Visible)	Visible Narrow	Yes	No
Thread Lines (Hidden)	Hidden	Yes	No
Thread Lines (Visible)	Hidden Narrow	Yes	No
View Edge (Hidden)	Hidden	Yes	Yes
View Edge (Visible)	Visible	Yes	Yes

In the view options dialog box, you can influence how Inventor reference parts are drawn. If you choose to display reference parts as a regular part, it is treated as a view edge, and drawn on the Visible or Hidden layers. If you choose to display the part as a reference part, it is treated as reference part and is drawn on the Reference layer. You acess the View Options dialog box using the View Options button on the Drawing View Editor ribbon contextual tab (VIEWEDIT command). You can also access it while creating a base view, from the Drawing View Creation ribbon contextual tab.

Layers are created on demand as and when drawing views are created. Furthermore, only the layers required to support the geometry in a view are created. For example, a view of a model space solid using the view style "Wireframe with visible edges only" creates only the Visible layer. For the same view, changing the visibility option to display tangent edges results in creation of the Visible Narrow layer.

Once a layer is created, it is not deleted automatically, even if the layer is no longer used. You must explicitly delete or purge unreferenced layers.

The default line type patterns and line weights are based on ISO 128-20:1996. You can however use the LAYER command and change these parameters.

If your company conventions require you to stick to specific layer names, you can rename the layers, after they are created. Subsequent views that you create use the renamed layers.

See also:

- Use Layers to Manage Complexity on page 516

Quick Reference

Commands

LAYER

Manages layers and layer properties.

VIEWEDIT

Edits an existing drawing view.

VIEWSETPROJ

Specifies the active project file for drawings containing drawing views from Inventor models.

Work Around Common Problems

Tips to handle some common issues that you are likely to face.

Empty Drawing Views

The VIEWBASE and VIEWPROJ commands do not allow the creation of empty drawing views. However, erasing the source geometry from model space or from the referenced Inventor model results in empty drawing views.

An empty drawing view is displayed as a 50 mm square. Its border is visible and is drawn in red.

Out-of-Date Inventor Models

An Inventor models can go out-of-date when changes are made directly to parts referenced by the model. This problem usually occur when parts are referenced in multiple assemblies.

The program does not prevent you from creating or editing drawing views derived from out-of-date Inventor models. It however displays a warning

message so that you are aware that the drawing views do not show the latest changes made to the source.

To ensure that the drawing views show the latest changes, use Autodesk Inventor to update the out-of-date Inventor model.

Unresolved Inventor File References

When files referenced by an Inventor model are not found automatically, drawing views derived from this model are said to contain unresolved file references.

A drawing view can contain unresolved file references for any of the following reasons.

a Paths to referenced files are not specified in the active project.

b The referenced files were renamed, moved, or deleted, and no longer exists within any of the storage locations defined by the active project.

c AutoCAD is pointing to the wrong active project file, or has not selected any project file as the active project file.

For cases a and b, use Autodesk Inventor to update the project file to specify the current location of the files that are not found automatically.

For case c, use the VIEWSETPROJ command to select the correct project file as the active project file.

Create Drawings from Other 3D CAD Models

Import IGES, CATIA®, Pro/ENGINEER®, STEP, SolidWorks®, JT, NX, Parasolid and Rhinoceros®(Rhino) files and generate drawings from them.

The IMPORT command lets you import a wide range of file formats into model space. The 3D CAD file formats you can import include the following:

- IGES (all versions)
- STEP (AP214 and AP203E2)
- Rhinoceros (Rhino)
- Pro/ENGINEER (up to Wildfire 5.0)
- CATIA V4 (all revisions)
- CATIA V5 (R10 - R19)

- Parasolid (up to V23)
- JT (7.0; 8.0; 8.1; 8.2; 9.0-9.5)
- NX
- SolidWorks (2003 - 2010)

After you import the 3D model, the model is available as an AutoCAD 3D model, in model space. Thereafter you generate drawings for the 3D model using the workflow for AutoCAD 3D models on page 1269.

Imported models are not associative with the source 3D model. Hence, drawings created in this manner do not update when the source 3D model changes.

See also:

- Import Other File Formats on page 1849

Quick Reference

Commands

IMPORT

Imports files of different formats into the current drawing.

Work with Sections

Create cross sections of 3D models.

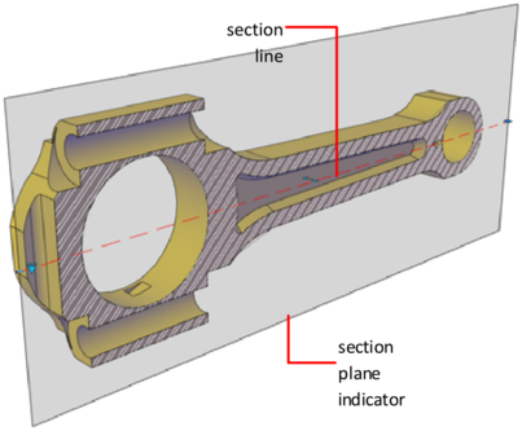
Overview of Section Objects

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.

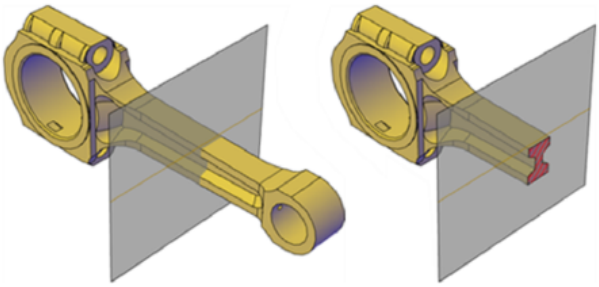


Store Properties in Section Lines

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

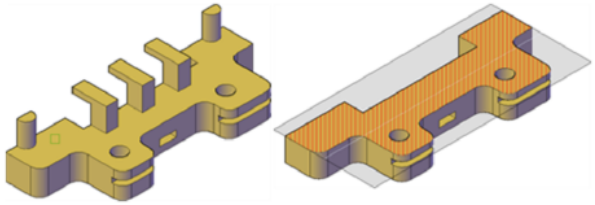
Create cross sections to show interior details of 3D objects.

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

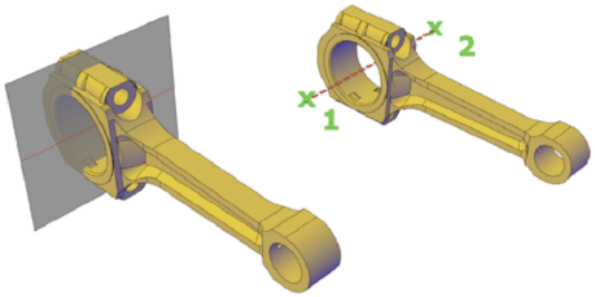
One way to set the section plane is to click the face of an existing 3D object. (As you move the cursor, a dotted outline indicates the side of the plane to be selected.) The section plane is automatically aligned to the plane of the face you select.



Section object aligned to face

Create a Straight Cutting Plane

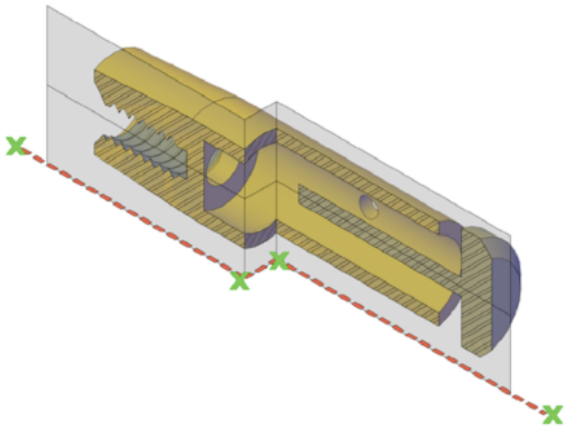
Pick two points to create a straight cutting plane.



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.

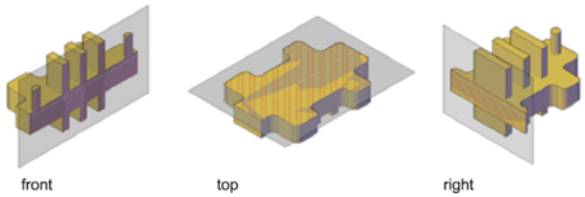
Create a section line that has jogged segments by using the Draw Section option of SECTIONPLANE to pick multiple points throughout the 3D model.



Section object with jogged segment

Create Orthographic Sections

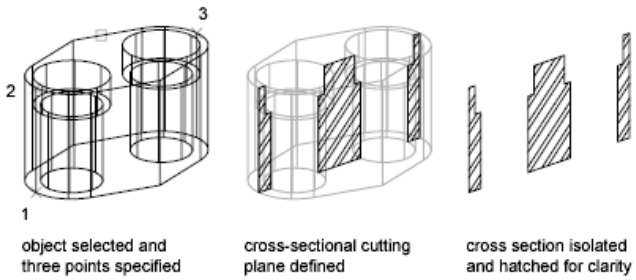
You can align section objects to a specified orthographic orientation of the current UCS, such as front, back, bottom, top, left, or right.



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.

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.

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.

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.

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.

Quick Reference

Commands

SECTIONPLANE

Creates a section object that acts as a cutting plane through 3D objects.

SECTION

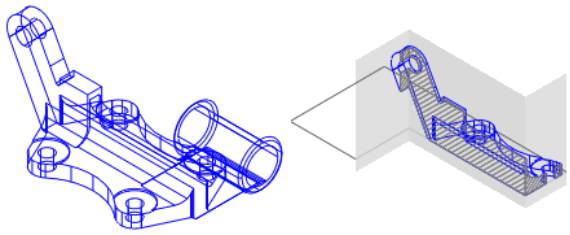
Uses the intersection of a plane and solids, surfaces, or mesh to create a region.

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 plane that has multiple segments (jogs), using the Draw Section option of the SECTIONPLANE command. You can also add a jog to an existing section plane by selecting the section you want to add a job to and then right-clicking. From the shortcut menu, click Add Jog to Section (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.

Quick Reference

Commands

SECTIONPLANE

Creates a section object that acts as a cutting plane through 3D objects.

SECTIONPLANEJOG

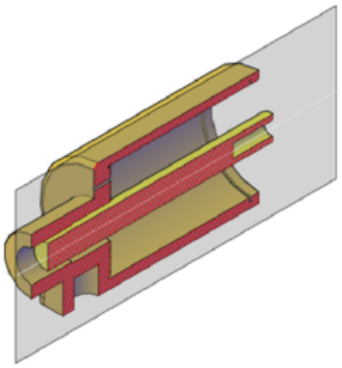
Adds a jogged segment to a section object.

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.

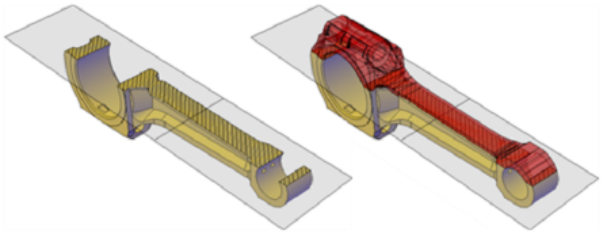


You can use live sectioning to analyze a model by moving the section object through the object. For example, sliding the section object through an engine assembly helps you visualize its internal components. You can use this method to create a cross section view that you can save or reuse.

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. 

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

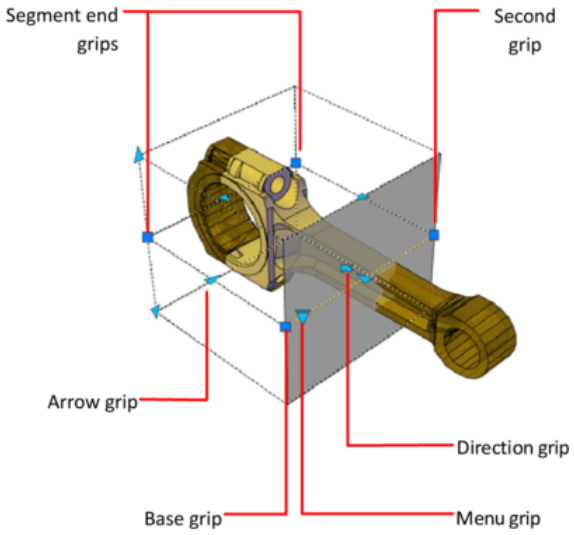
Creates a section object that acts as a cutting plane through 3D objects.

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**. 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. (Section Boundary and Volume states only.)
- **Segment end grips**. 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. (Section Boundary and Volume states only.)

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

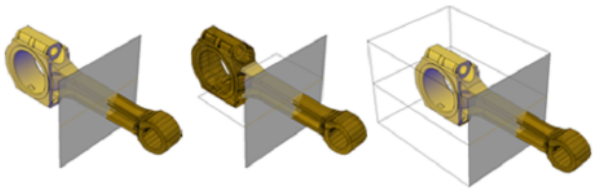
GRIPSIZE

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 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.

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.

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.

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.

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.

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 Named Views

Associate section objects with named views.

When you activate a named view 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. Later, you can restore a saved sectional view 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, you can quickly switch between the two views and see the desired cross section.

NOTE A camera is similar to a named view, with the exception of being able to manipulate the view seen through it using grip editing.

To associate a section object with a named view

1. Click View tab ➤ Views panel ➤ Named Views. 
2. In the View Manager, in the left pane, under Model Views, select a named view.
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. When the named view is restored, live sectioning is turned on for the associated section object.

Quick Reference

Commands

SECTIONPLANE

Creates a section object that acts as a cutting plane through 3D objects.

Save and Publish Section Objects

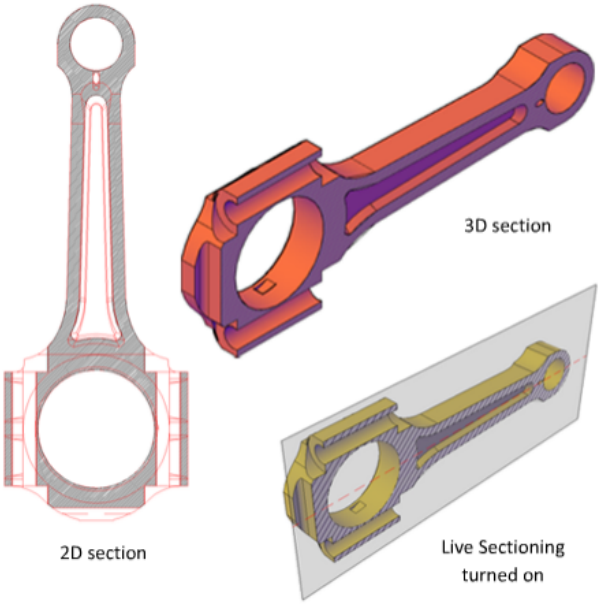
Save a section object as a block or tool, or publish it.

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 62

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.

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.

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

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.

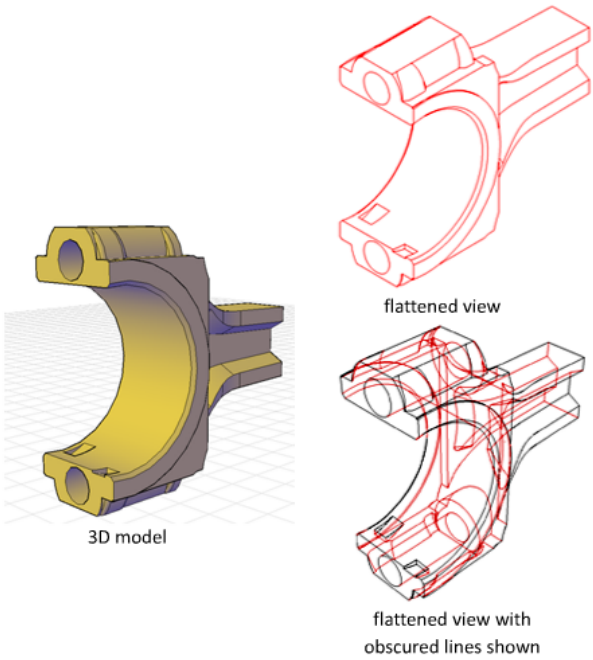
Create a Flattened View

Create a flattened view of the 3D solids and regions in the current view.

Create a 2D Presentation of a 3D Model

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 Block Created with Flatshot

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 drawing 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.

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.

SOLDRAW

Generates profiles and sections in layout viewports created with SOLVIEW.

SOLVIEW

Creates orthographic views, layers, and layout viewports automatically for 3D solids.

Part 9: Annotate Drawings

Work with Annotations 27

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.

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 on page 2289. 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 model or layout 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 on page 2289 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 238.

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 1334.
2. In model space, set the annotation scale to the scale at which the annotations will be plotted or displayed on page 1327.
3. Create annotative objects using annotative styles on page 1334.

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 1353.
2. Set the annotation scale to the new scale on page 1328 (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 431 or make a layout current on page 439.
2. Create viewports on page 448.
3. Set the annotation scale for each viewport on page 1328. (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 1351. For more information about adding scales to annotative objects, see Add and Modify Scale Representations on page 1353.

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.

Set Annotation Scale

Annotation scale on page 2289 is a setting that is saved with model space, layout viewports, and model views. When you add annotative on page 2289 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 layout or when a viewport is selected, the current annotation scale is displayed on the status bar. You can use the status bars to change the annotation scale. You can reset the annotation scale list to the default list of scales stored with your user profile in the Default Scale List dialog box.

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 status bar.

Use the CANNOSCALE system variable to set a default annotation scale setting.

You can reset the list of annotative scales in a drawing to the default list of either metric or imperial scales defined in the registry with the Default Scale dialog box. The unused scales in the drawing are purged and the customized list of scales from your user profile are merged into the drawing

See also:

- Drawing Status Bar on page 44

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

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.

Create Annotative Objects

Objects that are commonly used to annotate drawings have a property called Annotative on page 2289. When the Annotative property for these objects is turned on (set to Yes), these objects are called *annotative objects*

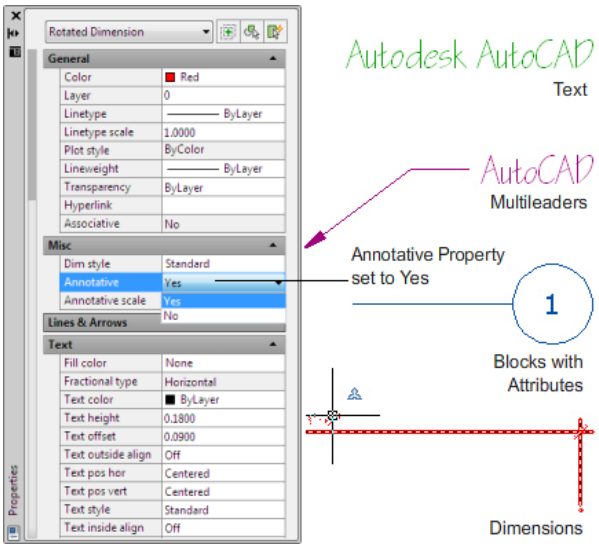
Overview of Creating Annotative Objects

When you add annotations to your drawing, you can turn on the Annotative on page 2289 property for those objects. These annotative objects are scaled based on the current annotation scale on page 2289 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.

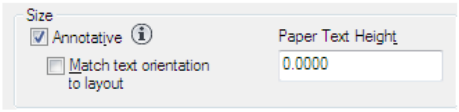
The following objects can be annotative (have an Annotative property):

- Hatches
- Text (single-line and multiline)
- Dimensions
- Tolerances
- Leaders and multileaders (created with MLEADER)
- Blocks
- Attributes

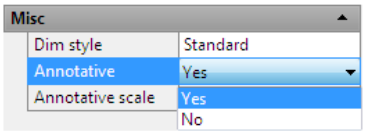


Many of the dialog boxes used to create these objects contain an Annotative check box where you can make the object annotative. You can also change existing objects to be annotative by changing the annotative property in the Properties palette.

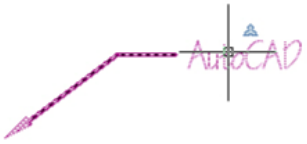
When you hover the cursor over an annotative object that supports one annotation scale, the cursor displays a  icon. When the object supports more than one annotation scale, it displays a  icon.



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 annotative on page 2289 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 drawing or DXF file formats.

Annotative objects may have multiple scale representations on page 2318. When visual fidelity is on, annotative objects are decomposed and scale representations are saved (in an anonymous block on page 2289) 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 layout. More annotation objects may be displayed on the Model layout 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.

See also:

- Work with Annotative Styles on page 1333

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

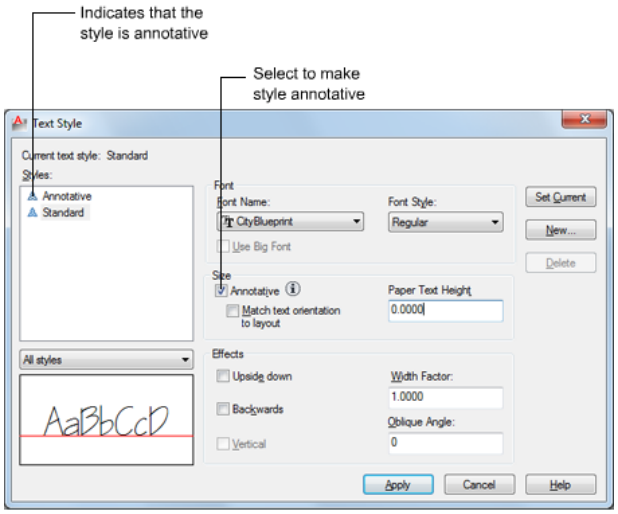
Creates a single-line text object.

Work with Annotative Styles

You can minimize the steps to annotate a drawing by using annotative styles.

Annotative text, dimension, and multileader styles create annotative on page 2289 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 Text Height value for any annotative text styles you create. The Paper Text Height setting specifies the height of the text in paper space.

NOTE If you've specified the Text Height value for a dimension or multileader style, this setting overrides the text style Paper Text Height setting.

If you redefine styles to be annotative or non-annotative, 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 non-annotative), the object's annotative properties will match

that of the new style. If the style does not have a fixed height (the Text 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 1469
- Create Annotative Text on page 1335
- Use Dimension Styles on page 1530
- Create Annotative Dimensions and Tolerances on page 1339
- Work with Leader Styles on page 1451
- Create Annotative Leaders and Multileaders on page 1344

To create an annotative style

- Follow the steps in one of the following procedures
  - To create a new annotative text style on page 1336
  - To change an existing non-annotative text style to annotative on page 1336
  - To create a new annotative dimension style on page 1340
  - To change an existing dimension style to annotative on page 1341
  - To create a new annotative multileader style on page 1344
  - To change an existing multileader style to annotative on page 1345

To create annotative objects from annotative styles

- Follow the steps in one of the following procedures
  - To create annotative single-line text on page 1337
  - To create annotative multiline text on page 1337
  - To create an annotative dimension on page 1341
  - To create an annotative multileader on page 1345

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.

Create Annotative Text

Use annotative on page 2289 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 annotation scale on page 2289 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 non-annotative text to annotative by changing the text's Annotative property to Yes (On). 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 1356.

See also:

- Create Text on page 1402
- Work with Annotative Styles on page 1333

■ Set Orientation for Annotations on page 1356

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.

To change an existing non-annotative 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.

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.

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.

To change existing multiline text to annotative or non-annotative

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 non-annotative.

   When the Annotative button is depressed, the text is annotative. When the button is not depressed, the text is non-annotative.
3. Click OK to save the changes.

To change existing text (single-line or multiline) to be annotative or non-annotative

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.

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**.

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.

Quick Reference

Commands

MTEXT

Creates a multiline text object.

STYLE

Creates, modifies, or specifies text styles.

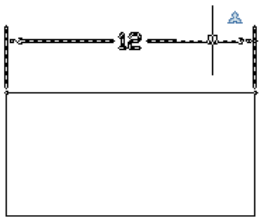
TEXT

Creates a single-line text object.

Create Annotative Dimensions and Tolerances

You can create annotative on page 2289 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 annotation scale on page 2289.



If you associate a dimension to an annotative object, the associativity of the dimension is lost.

You can also change an existing non-annotative dimension to annotative by changing the dimension's Annotative property to Yes (On).

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 1523
- Use Dimension Styles on page 1530
- Work with Annotative Styles on page 1333

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.

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.

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. If an annotative style does not exist, you will need to create

one first. For information on creating an annotative dimension style see, To create a new annotative dimension style on page 1340.

3 Click Set Current.

4 Click Close.

5 Click Annotate tab ➤ Dimensions panel ➤ Dimension drop-down list ➤ select a dimension type, or at the Command prompt, enter a dimension command.

If the Select Annotation Scale dialog box is displayed, select the default scale you want to use for the new dimension object and click OK.

6 Place the dimension in the drawing area.

7 Select the new dimension and right-click. Click Annotative Object Scale ➤ Add/Delete Scales.

8 In the Annotation Object Scale dialog box, click Add.

9 In the Add Scales to Object dialog box, select the object scales you want to assign to the dimension. Click OK.

10 In the Annotation Object Scale dialog box, click OK.

To change an existing dimension to annotative or non-annotative

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.

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**.

To create an annotative tolerance

1 Follow the steps in To create geometric tolerances on page 1622.

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.

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.

Create Annotative Leaders and Multileaders

Leaders and multileaders on page 2309 are used to add call outs to your drawings. You can create annotative on page 2289 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 1439
- Work with Leader Styles on page 1451

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.

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.

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.

To change an existing leader or multileader to annotative or non-annotative

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.

Quick Reference

Commands

MLEADERSTYLE

Creates and modifies multileader styles.

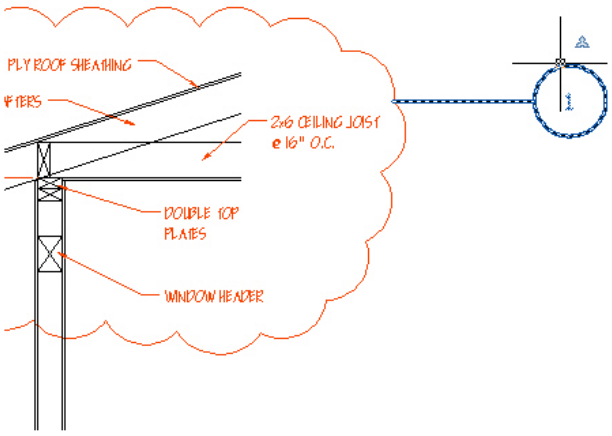
STYLE

Creates, modifies, or specifies text styles.

Create Annotative Blocks and Attributes

If you want to use geometric objects to annotate your drawing, combine the objects into an annotative block definition.

Annotative on page 2289 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 layout with the annotation scale on page 2289 set to 1:1.

When creating and working with annotative blocks and annotative objects within blocks, the following points should be noted:

- Non-annotative 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 non-annotative blocks. Use annotative attributes with non-annotative 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 Text 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 1356.

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 annotative on page 2289 blocks into a drawing.

See also:

- Work with Blocks on page 915
- Attach Data to Blocks (Block Attributes) on page 939
- Set Orientation for Annotations on page 1356

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.

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.

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.

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.

Create Annotative Hatches

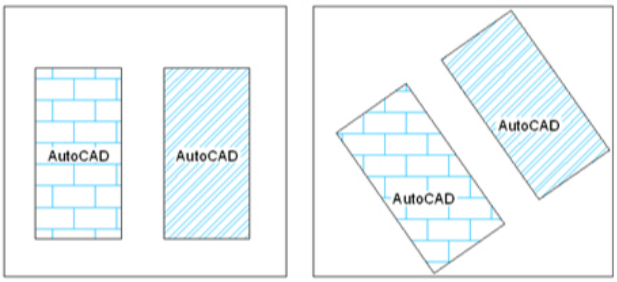
Use an annotative hatch to symbolically represent material such as sand, concrete, steel, earth, etc.

An annotative on page 2289 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.

You can use the HPANNOTATIVE system variable to specify whether or notnew hatches are annotative. By default, new hatch objects are non-annotative.

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 At the command prompt, enter settings.

3 In the Hatch and Gradient dialog box, click Add: Select Objects.

4 Specify the object or objects you want to hatch.

5 Under Options, select Annotative.

6 Click OK.

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

HPANNOTATIVE

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 44

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

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.

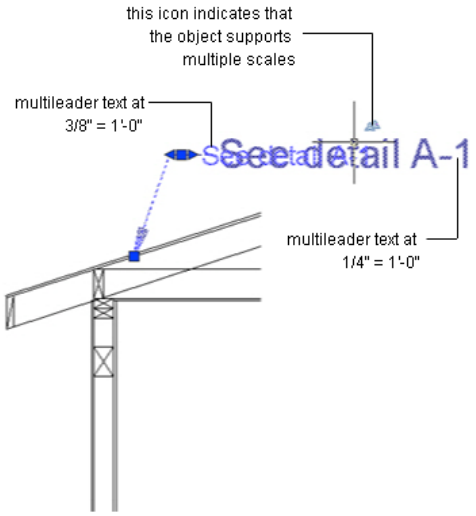
Add and Modify Scale Representations

When you create an annotative on page 2289 object in your drawing, it supports one annotation scale on page 2289, 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 scale representations on page 2318 to the object.

For example, if an annotative multileader supports two annotation 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.

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.

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.

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.

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

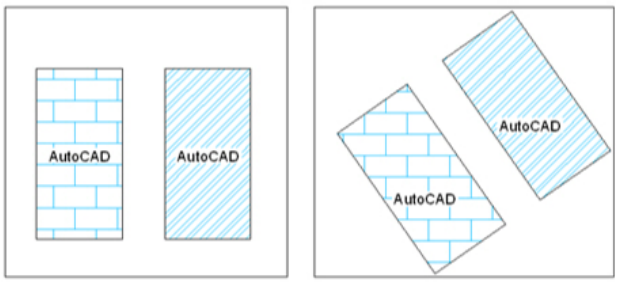
SELECTIONANNODISPLAY

Controls whether alternate scale representations are temporarily displayed in a dimmed state when an annotative object is selected.

Set Orientation for Annotations

Annotative on page 2289 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 1469
- Create Annotative Text on page 1335
- Create Annotative Blocks and Attributes on page 1346
- Create Annotative Hatches on page 1350

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.

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.

To match the layout's orientation for an existing annotative text object

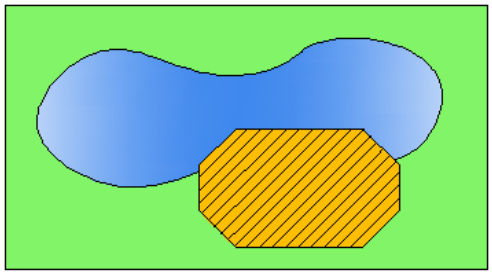
1. In the drawing, select an annotative text object.
2. Click Annotate tab ➤ Text panel ➤ Text Style. 
3. In the Text Style dialog box, under Size, select Match Orientation to Layout.

Hatches, Fills, and Wipeouts

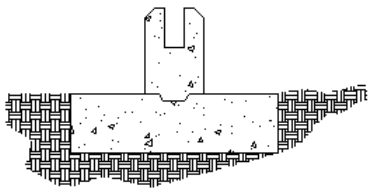
28

- Overview of Hatch Patterns and Fills on page 1359
- Specify Hatch and Fill Areas on page 1367
- Control the Appearance of Hatches on page 1372
- Modify Hatches and Fills on page 1389
- Create a Blank Area to Cover Objects on page 1395

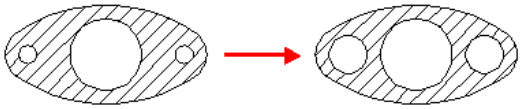
Overview of Hatch Patterns and Fills



Hatches and fills do not have to be bounded. In the following illustration, the concrete hatches are *bounded*, while the earth hatches are *unbounded*.

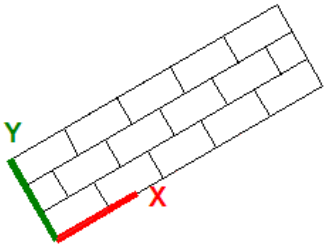


By default, bounded hatches are *associative*, which means that the hatch object is associated with the hatch boundary objects, and changes to the boundary objects are automatically applied to the hatch.



To maintain associativity, the boundary objects must continue to completely enclose the hatch.

The alignment and orientation of a hatch pattern is determined by the current location and orientation of the user coordinate system, in addition to controls in the user interface.



Moving or rotating the UCS is an alternate method for controlling hatch patterns.

NOTE By default, a preview of the hatch displays as you move the cursor over enclosed areas. To improve the response time in large drawings, turn off the hatch preview feature with the HPQUICKPREVIEW system variable, or decrease the time before the preview is temporarily canceled with the HPQUICKPREVTIMEOUT system variable.

Alternatively, solid-filled areas can be created using

- 2D solids (SOLID)
- Wide polylines (PLINE)
- Donuts (DONUT)
- Traces (TRACE)

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.

2. On the DesignCenter toolbar, click Search.
3. In the Search dialog box, do the following:
   - From the Look For drop-down list, select Hatch Pattern Files.
   - From the In drop-down list, select the drive where the program is installed.
   - Confirm that Search Subfolders is selected.
   - On the Hatch Pattern Files tab, 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 in DesignCenter folders list.

5. In the search results, double-click the filename 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 on the Clipboard.
   - **Create Tool Palette.** Creates a new tool palette that contains the selected pattern.
7. From the content area of DesignCenter, drag a hatch pattern into an enclosed area 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 selecting it to display the Hatch Editor tab. If the ribbon is closed, use the Properties palette or HATCHEDIT.

To hatch or fill areas (ribbon turned on)

1. Click Home tab ➤ Draw panel ➤ Hatch. 
2. On the Properties panel, select one of the options from the Hatch Type drop-down list.
3. On the Pattern panel, click a hatch pattern or fill.
4. Specify a point inside each area to be hatched.
5. On the ribbon, make any adjustments as needed.

   On the Properties panel, you can change the hatch type and colors or modify the transparency level, angle, or scale for the hatch.
6. (Optional) Expand the Options panel, and select one of the draw order options from the bottom drop-down list.

   You can change the draw order of the hatch so that the hatch is displayed either behind or in front of the hatch boundary, or behind or in front of all other objects.
7. Press Enter, or click Close on the ribbon to apply the hatch and exit the command.

To hatch or fill areas (ribbon turned off)

1. Start the Hatch command from a menu or toolbar. 
2. In the Hatch and Gradient dialog box, under Boundaries, click Add: Pick Points.
3. Specify a point inside each area that you want to hatch, and press Enter.
4. In the Hatch and Gradient dialog box, select a pattern or Solid from the Pattern list.

   To see how the hatch pattern will look, click the Preview button. Press Esc when you finish previewing to return to the dialog box.

5 In the Hatch and Gradient dialog box, make adjustments, if necessary.

   To access additional options, click the More Options button at the bottom-right of the dialog box.

6 (Optional) From the Draw Order drop-down list, choose one of the options.

   You can change the draw order of the hatch so that the hatch is displayed either behind or in front of the hatch boundary, or behind or in front of all other objects.

7 Click OK to apply the hatch.

To hatch selected objects (ribbon turned on)

1 Click Home tab ➤ Draw panel ➤ Hatch. 

2 On the Boundaries panel, click Select.

3 Select the objects that you want to hatch.

4 Press Enter, or click Close on the ribbon to apply the hatch and exit the command.

To hatch selected objects (ribbon turned off)

1 Start the Hatch command from a menu or toolbar. 

2 In the Hatch and Gradient dialog box, under Boundaries, click Add: Select Objects.

3 Select the objects that you want to hatch and press Enter.

4 Click OK to apply the hatch.

Quick Reference

Commands

ADCENTER

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

GRADIENT

Fills an enclosed area or selected objects with a gradient fill.

HATCH

Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

HATCHEDIT

Modifies an existing hatch or fill.

HATCHTOBACK

Sets the draw order for all hatches in the drawing to be behind all other objects.

MATCHPROP

Applies the properties of a selected object to other objects.

PROPERTIES

Controls properties of existing objects.

UCS

Sets the origin and orientation of the current UCS.

System Variables

GFANG

Specifies the angle of a gradient fill.

GFCLR1

Specifies the color for a one-color gradient fill or the first color for a two-color gradient fill.

GFCLR2

Specifies the second color for a two-color gradient fill.

GFCLRLUM

Controls the tint or shade level in a one-color gradient fill.

GFCLRSTATE

Specifies whether a gradient fill uses one color or two colors.

GFNAME

Specifies the pattern of a gradient fill.

GFSHIFT

Specifies whether the pattern in a gradient fill is centered or is shifted up and to the left.

HPANG

Sets the angle for new hatch patterns.

HPANNOTATIVE

Controls whether a new hatch pattern is annotative.

HPASSOC

Controls whether hatches and fills are associative.

HPBACKGROUNDCOLOR

Controls the background color for hatch patterns.

HPBOUND

Controls the object type created by HATCH and BOUNDARY.

HPBOUNDRETAIN

Controls whether boundary objects are created for new hatches and fills.

HPCOLOR

Sets a default color for new hatches.

HPDLGMODE

Controls the display of the Hatch and Gradient dialog box and the Hatch Edit dialog box.

HPDOUBLE

Specifies hatch pattern doubling for user-defined patterns.

HPDRAWORDER

Controls the draw order of hatches and fills.

HPGAPTOL

Treats a set of objects that almost enclose an area as a closed hatch boundary.

HPINHERIT

Controls whether to inherit the hatch origin when using the Inherit Properties option in HATCH and HATCHEDIT.

HPISLANDDETECTION

Controls how islands within the hatch boundary are treated.

HPISLANDDETECTIONMODE

Controls whether internal closed boundaries, called islands, are detected.

HPLAYER

Specifies a default layer for new hatches and fills.

HPMAXAREAS

Sets the maximum number of enclosed areas that a single hatch object can have and still automatically switch between solid and pattern hatches during zoom operations.

HPMAXLINES

Sets the maximum number of hatch lines that are generated in a hatch operation.

HPNAME

Sets the default hatch pattern name.

HPOBJWARNING

Sets the number of hatch boundary objects that can be selected before displaying a warning message.

HPORIGIN

Sets the hatch origin point for new hatch patterns relative to the current user coordinate system.

HPORIGINMODE

Controls how the default hatch origin point is determined.

HPQUICKPREVIEW

Controls whether a hatch preview is displayed when specifying a hatch area.

HPQUICKPREVTIMEOUT

Sets the maximum time for a hatch preview to generate before the preview is automatically cancelled.

HPSCALE

Sets the hatch pattern scale factor.

HPSEPARATE

Controls whether a single hatch object or separate hatch objects are created when operating on several closed boundaries.

HPSPACE

Sets the hatch pattern line spacing for user-defined patterns.

HPTRANSPARENCY

Sets the default transparency for new hatches and fills.

MIRRHATCH

Controls how MIRROR reflects hatch patterns.

PICKSTYLE

Controls the use of group selection and associative hatch selection.

Specify Hatch and Fill Areas

Define boundaries for hatches and fills from existing objects or from specified boundary points.

Use one of several methods to specify the 2D geometric boundaries of a hatch or fill.

- Specify a point in an area that is enclosed by objects.
- Select objects that enclose an area.
- Specify boundary points using the Draw option of -HATCH.
- Drag a hatch into an enclosed area from a tool palette or DesignCenter.

NOTE Enclosed areas can be hatched only if they are in a plane parallel to the *XY* plane of the current UCS.

Create Associative Hatches

Associative hatches are automatically updated when their boundary objects are modified. Minor changes in the boundary of an associative hatch do not require erasing and re-creating the hatch.



Hatch associativity is turned on by default and is controlled by the HPASSOC system variable. You can also control hatch associativity using the following tools in the user interface:

- Hatch and Gradient dialog box
- Hatch Edit dialog box
- Hatch panel on the ribbon
- Properties palette

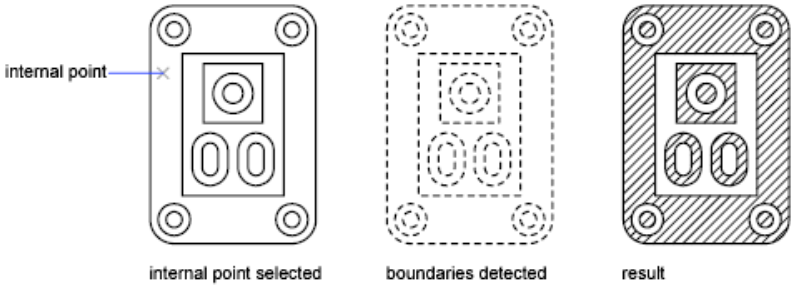
Nonassociative hatches are not updated when their original boundary is changed.

Hatch Enclosed Areas Within Boundaries

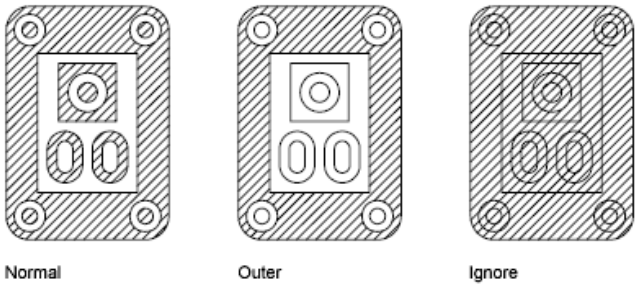
Enclosed areas within hatch boundaries are called *islands*. There are four island detection styles available from the user interface:

- Normal island detection
- Outer island detection (recommended)
- Ignore island detection
- No island detection (legacy behavior that is similar to the Ignore style)

Using Normal island detection, if you specify the internal pick point shown, islands remain unhatched and islands within islands are hatched.



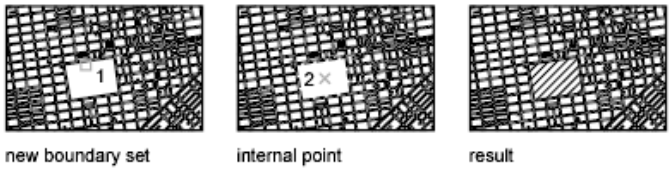
Using the same pick point, the results of the options are compared below.



NOTE Text objects are treated as islands. If island detection is turned on, the result always leaves a rectangular space around the text.

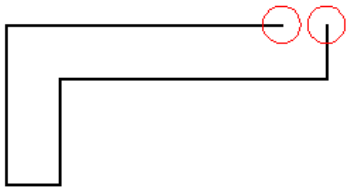
Include Objects in a Boundary Set

When hatching a small area in a large, complex drawing, you can save time by selecting a smaller set of objects in the drawing to be used in determining the hatch boundary.



Identify Gaps in Hatch Boundaries

If the specified internal point is not within a fully enclosed area, red circles are displayed at the unconnected endpoints of the boundary to identify the gaps.



The red circles remain displayed after you exit HATCH. They are removed when you specify another internal point for the hatch, or when you use REDRAW, REGEN, or REGENALL.

To hatch an area whose boundary is not quite closed, do one of the following:

- Locate the gaps and modify the boundary objects so they form a closed boundary.
- Set the HPGAPTOL system variable to a value large enough to bridge the gaps. HPGAPTOL applies only to gaps between geometric objects that would meet if extended.

NOTE By default, a preview of the hatch displays as you move the cursor over bounded areas. To improve the response time in large drawings, turn off the hatch preview feature (HPQUICKPREVIEW system variable), or decrease the preview timeout value (HPQUICKPREVTIMEOUT system variable).

See also:

- Reshape a Hatch or Fill on page 1392

To define a boundary set in a complex drawing (ribbon turned on)

1. Click Home tab ➤ Draw panel ➤ Hatch. 
2. Expand the Boundaries panel, and click the Select New Boundary Set button at the bottom-left of the expanded panel.
3. Select the objects to be considered for the hatch and press Enter.

   Selecting the objects with a *crossing window* is often the best method.
4. Specify a point inside each area that you want hatched.
5. Press Enter, or click Close on the ribbon to apply the hatch and exit the command.

To define a boundary set in a complex drawing (ribbon turned off)



1. Start the Hatch command from a menu or toolbar.
2. In the Hatch and Gradient dialog box, click the More Options button at the bottom-right.
3. Under Boundary Set, click New.
4. Select the objects to be considered for the hatch and press Enter.
   Selecting the objects with a *crossing window* is often the best method.
5. In the Hatch and Gradient dialog box, click Add: Pick Points.
6. Specify a point inside each area that you want to hatch.
7. Click OK to apply the 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

HPANG

Sets the angle for new hatch patterns.

HPASSOC

Controls whether hatches and fills are associative.

HPBOUND

Controls the object type created by HATCH and BOUNDARY.

HPBOUNDRETAIN

Controls whether boundary objects are created for new hatches and fills.

HPGAPTOL

Treats a set of objects that almost enclose an area as a closed hatch boundary.

HPISLANDDETECTION

Controls how islands within the hatch boundary are treated.

HPISLANDDETECTIONMODE

Controls whether internal closed boundaries, called islands, are detected.

HPQUICKPREVIEW

Controls whether a hatch preview is displayed when specifying a hatch area.

HPQUICKPREVTIMEOUT

Sets the maximum time for a hatch preview to generate before the preview is automatically cancelled.

HPSEPARATE

Controls whether a single hatch object or separate hatch objects are created when operating on several closed boundaries.

Control the Appearance of Hatches

Specify a hatch pattern or fill, and control its alignment and scale.

Choose a Hatch Pattern or Fill

Choose from three types of hatch patterns, and two types of fills.

- **Predefined hatch patterns.** Choose from over 70 ANSI, ISO, and other industry-standard hatch patterns that are available. You can also use hatch patterns from hatch pattern libraries supplied by other companies. Hatch patterns are defined in the *acad.pat* and *acadiso.pat* files.
- **User-defined hatch patterns.** Define a hatch pattern that uses the current linetype with a specified spacing and angle.
- **Custom hatch patterns.** Define a custom hatch pattern definition in a *.pat* file.
- **Solid fill.** Fill an area with a solid color by choosing the SOLID predefined hatch.

- **Gradient fill.** Fill an enclosed area with a color gradient. A gradient fill can be displayed as a *tint* (a color mixed with white), a *shade* (a color mixed with black), or a smooth transition between two colors.



Gradients that mimic colors displayed on a cylinder, a sphere, or other shapes are available.

NOTE You cannot use plot styles to control the plotted color of gradient fills.

Assign a Background Color to Hatch Patterns

Predefined, user defined, and custom hatch patterns, can be assigned a background fill color. The background fill color shares the same level of transparency as the pattern itself.

See also:

- Modify Hatch Properties on page 1389
- "Overview of Hatch Pattern Definitions" in the *Customization Guide*

To create a hatch with a predefined hatch pattern (ribbon turned on)

1. Click Home tab ➤ Draw panel ➤ Hatch. 
2. On the Properties panel, click Pattern from the Hatch Type drop-down list.
3. (Optional) On the Properties panel, click a color override from the Hatch Color drop-down list. Specify any other options as needed.
4. On the Pattern panel, click a hatch pattern.
5. Specify a point inside each area that you want hatched.
6. Press Enter, or click Close on the ribbon to apply the hatch and exit the command.

To create a hatch with a predefined hatch pattern (ribbon turned off)

1. Start the Hatch command from a menu or toolbar. 
2. In the Hatch and Gradient dialog box, Hatch tab, under Type and Pattern, click Predefined from the Type drop-down list.
3. Select a pattern from the Pattern drop-down list.
4. (Optional) Select a color override from the Color drop-down list. Specify any other options as needed.
5. Under Boundaries, click Add: Pick Points.
6. Specify a point inside each area that you want to hatch and press Enter.
7. Click OK to apply the hatch.

To fill an area with a solid color (ribbon turned on)

1. Click Home tab ➤ Draw panel ➤ Hatch. 
2. On the Properties panel, click Solid from the Hatch Type drop-down list, or click Solid on the Pattern panel.
3. (Optional) On the Properties panel, click a color override from the Hatch Color drop-down list. Specify any other options as needed.
4. Specify a point inside each area that you want to fill.
5. Press Enter, or click Close on the ribbon to apply the hatch and exit the command.

To fill an area with a solid color (ribbon turned off)

1. Start the Hatch command from a menu or toolbar. 
2. In the Hatch and Gradient dialog box, Hatch tab, under Type and Pattern, click Predefined from the Type drop-down list.
3. Click Solid from the Pattern drop-down list.
4. (Optional) Select a color override from the Color drop-down list. Specify any other options as needed.

5 Under Boundaries, click Add: Pick Points.

6 Specify a point inside each area that you want to hatch and press Enter.

7 Click OK to apply the hatch.

Quick Reference

Commands

GRADIENT

Fills an enclosed area or selected objects with a gradient fill.

HATCH

Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

HATCHEDIT

Modifies an existing hatch or fill.

HATCHTOBACK

Sets the draw order for all hatches in the drawing to be behind all other objects.

MATCHPROP

Applies the properties of a selected object to other objects.

PROPERTIES

Controls properties of existing objects.

UCS

Sets the origin and orientation of the current UCS.

System Variables

GFANG

Specifies the angle of a gradient fill.

GFCLR1

Specifies the color for a one-color gradient fill or the first color for a two-color gradient fill.

GFCLR2

Specifies the second color for a two-color gradient fill.

GFCLRLUM

Controls the tint or shade level in a one-color gradient fill.

GFCLRSTATE

Specifies whether a gradient fill uses one color or two colors.

GFNAME

Specifies the pattern of a gradient fill.

GFSHIFT

Specifies whether the pattern in a gradient fill is centered or is shifted up and to the left.

HPANG

Sets the angle for new hatch patterns.

HPANNOTATIVE

Controls whether a new hatch pattern is annotative.

HPASSOC

Controls whether hatches and fills are associative.

HPBACKGROUNDCOLOR

Controls the background color for hatch patterns.

HPBOUND

Controls the object type created by HATCH and BOUNDARY.

HPBOUNDRETAIN

Controls whether boundary objects are created for new hatches and fills.

HPCOLOR

Sets a default color for new hatches.

HPDOUBLE

Specifies hatch pattern doubling for user-defined patterns.

HPDRAWORDER

Controls the draw order of hatches and fills.

HPINHERIT

Controls whether to inherit the hatch origin when using the Inherit Properties option in HATCH and HATCHEDIT.

HPISLANDDETECTION

Controls how islands within the hatch boundary are treated.

HPISLANDDETECTIONMODE

Controls whether internal closed boundaries, called islands, are detected.

HPLAYER

Specifies a default layer for new hatches and fills.

HPMAXAREAS

Sets the maximum number of enclosed areas that a single hatch object can have and still automatically switch between solid and pattern hatches during zoom operations.

HPMAXLINES

Sets the maximum number of hatch lines that are generated in a hatch operation.

HPNAME

Sets the default hatch pattern name.

HPOBJWARNING

Sets the number of hatch boundary objects that can be selected before displaying a warning message.

HPINHERIT

Controls whether to inherit the hatch origin when using the Inherit Properties option in HATCH and HATCHEDIT.

HPORIGIN

Sets the hatch origin point for new hatch patterns relative to the current user coordinate system.

HPORIGINMODE

Controls how the default hatch origin point is determined.

HPQUICKPREVIEW

Controls whether a hatch preview is displayed when specifying a hatch area.

HPQUICKPREVTIMEOUT

Sets the maximum time for a hatch preview to generate before the preview is automatically cancelled.

HPSCALE

Sets the hatch pattern scale factor.

HPSEPARATE

Controls whether a single hatch object or separate hatch objects are created when operating on several closed boundaries.

HPSPACE

Sets the hatch pattern line spacing for user-defined patterns.

HPTRANSPARENCY

Sets the default transparency for new hatches and fills.

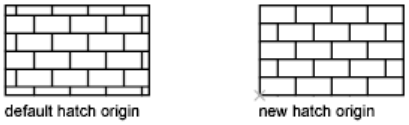
MIRRHATCH

Controls how MIRROR reflects hatch patterns.

Control the Hatch Origin Point

Each hatch pattern is aligned with an *origin point*. Changing the origin point shifts the pattern.

By default, hatch patterns are aligned with the origin point of the user coordinate system. However, sometimes you need to move the origin point of the hatch object. For example, if you create a brick pattern, you can start with a complete brick in the lower-left corner of the hatched area by specifying a new origin point.



The hatch origin and its behavior depend on settings in the user interface that control the HPORIGIN, HPORIGINMODE, and HPINHERIT system variables. Alternatively, you can control hatch patterns by changing the location and orientation of the user coordinate system.

See also:

- Modify Hatch Alignment, Scale, and Rotation on page 1391

To specify the alignment of a hatch pattern (ribbon turned on)



1. Click Home tab ➤ Draw panel ➤ Hatch.
2. On the Properties panel, click Pattern from the Hatch Type drop-down list.
3. On the Pattern panel, click a hatch pattern.
4. On the Origin panel, click Set Origin, and specify a point in the drawing.
   The hatch pattern will be aligned to this point.
5. On the Properties panel, specify a hatch angle.
6. Specify a point inside each area that you want to hatch.
7. Press Enter, or click Close on the ribbon to apply the hatch and exit the command.

Alternatively, you can control the origin and rotation of a hatch pattern by changing the origin and rotation of the user coordinate system (UCS) before you create the hatch.

To specify the alignment of a hatch pattern (ribbon turned off)



1. Start the Hatch command from a menu or toolbar.
2. In the Hatch and Gradient dialog box, Hatch tab, under Type and Pattern, click Predefined from the Type drop-down list.
3. Select a hatch pattern from the Pattern drop-down list.
4. Under Hatch Origin, click Specified Origin and click Click to Set New Origin. Specify a point in the drawing.
   The hatch pattern will be aligned to this point.
5. Under Angle and Scale, specify a hatch angle to rotate the hatch pattern.
6. Under Boundaries, click Add: Pick Points.
7. Specify a point inside each area that you want to hatch and press Enter.

8 Click OK to apply the hatch.

Alternatively, you can control the origin and rotation of a hatch pattern by changing the origin and rotation of the user coordinate system (UCS) before you create the 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.

PROPERTIES

Controls properties of existing objects.

UCS

Sets the origin and orientation of the current UCS.

System Variables

HPINHERIT

Controls whether to inherit the hatch origin when using the Inherit Properties option in HATCH and HATCHEDIT.

HPORIGIN

Sets the hatch origin point for new hatch patterns relative to the current user coordinate system.

HPORIGINMODE

Controls how the default hatch origin point is determined.

HPQUICKPREVIEW

Controls whether a hatch preview is displayed when specifying a hatch area.

HPQUICKPREVTIMEOUT

Sets the maximum time for a hatch preview to generate before the preview is automatically cancelled.

HPSCALE

Sets the hatch pattern scale factor.

HPSEPARATE

Controls whether a single hatch object or separate hatch objects are created when operating on several closed boundaries.

HPSPACE

Sets the hatch pattern line spacing for user-defined patterns.

Control the Scale of Hatch Patterns

The scale of hatch patterns can be set individually, or it can be set automatically based on the scale of each layout viewport.

- If you create hatch patterns exclusively for a single view or at a constant scale, you can set the current hatch scale manually in the interface or with the HPSCALE system variable.
- If you work with layout viewports in different scales, you can apply scale factors automatically by making them annotative. This method is more efficient than creating duplicate hatch pattern objects with different scale factors. For more information about using annotative scaling, see Create Annotative Hatches on page 1350.

NOTE To prevent accidental creation of an enormous number of hatch lines, the maximum number of hatch lines created in a single hatch operation is limited. This limit prevents memory and performance problems. However, you can change the maximum number of hatch lines with the HPMAXLINES system variable. Similarly the number of enclosed areas in single hatch is limited by the HPMAXAREAS system variable.

See also:

- Scale Annotations on page 1324
- Modify Hatch Alignment, Scale, and Rotation on page 1391
- Create Annotative Hatches on page 1350

To set the scale of a hatch pattern (ribbon turned on)

1. Click Home tab ➤ Draw panel ➤ Hatch. 
2. On the Properties panel, click Pattern from the Hatch Type drop-down list.
3. On the Pattern panel, click a hatch pattern.
4. On the Properties panel, enter a hatch pattern scale.
5. Specify a point inside each area that you want to hatch.
6. Press Enter, or click Close on the ribbon to apply the hatch and exit the command.

To set the scale of a hatch pattern (ribbon turned off)

1. Start the Hatch command from a menu or toolbar. 
2. In the Hatch and Gradient dialog box, Hatch tab, under Type and Pattern, click Predefined from the Type drop-down list.
3. Select a pattern from the Pattern drop-down list.
4. Under Angle and Scale, enter a new scale value or click the drop-down arrow to display a list of scales.
5. Under Boundaries, click Add: Pick Points.
6. Specify a point inside each area that you want to hatch and press Enter.
7. Click OK to apply the 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.

MATCHPROP

Applies the properties of a selected object to other objects.

PROPERTIES

Controls properties of existing objects.

System Variables

HPANNOTATIVE

Controls whether a new hatch pattern is annotative.

HPINHERIT

Controls whether to inherit the hatch origin when using the Inherit Properties option in HATCH and HATCHEDIT.

HPMAXAREAS

Sets the maximum number of enclosed areas that a single hatch object can have and still automatically switch between solid and pattern hatches during zoom operations.

HPMAXLINES

Sets the maximum number of hatch lines that are generated in a hatch operation.

HPOBJWARNING

Sets the number of hatch boundary objects that can be selected before displaying a warning message.

HPINHERIT

Controls whether to inherit the hatch origin when using the Inherit Properties option in HATCH and HATCHEDIT.

HPSCALE

Sets the hatch pattern scale factor.

HPSPACE

Sets the hatch pattern line spacing for user-defined patterns.

Set Property Overrides for Hatches and Fills

Control the default color, layer, and transparency of hatch objects separately from other objects.

Hatch objects have an additional capability that is not available with other types of objects. You can specify which layer, color, and transparency settings will be automatically applied to each new hatch object, regardless of the *current* property settings. This can save you time.

For example, you can specify that all new hatch objects are automatically created on a specified layer regardless of the current layer setting.

NOTE If you do not want to override the current property settings, select Use Current for the hatch's layer, color, and transparency settings.

See also:

- Modify Hatch Properties on page 1389
- Control How Overlapping Objects Are Displayed on page 587

To set a transparency override for new hatches (ribbon turned on)

1. Click Home tab ➤ Draw panel ➤ Hatch. 
2. On the Properties panel, drag the Hatch Transparency slider or enter a value.
3. Specify a point inside each area that you want to hatch.
4. Press Enter, or click Close on the ribbon to apply the hatch and exit the command.

All new *hatch* objects will use this transparency value instead of the current transparency value used by all *other* objects.

To set a transparency override for new hatches (ribbon turned off)

1. Start the Hatch command from a menu or toolbar. 
2. In the Hatch and Gradient dialog box, under Options, select Specify Value from the Transparency drop-down list.
3. Enter transparency value or drag the slider.
4. Under Boundaries, click Add: Pick Points.
5. Specify a point inside each area that you want to hatch and press Enter.

6 Click OK to apply the hatch.

All new *hatch* objects will use this transparency value instead of the current transparency value used by all *other* objects.

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.

HATCHTOBACK

Sets the draw order for all hatches in the drawing to be behind all other objects.

MATCHPROP

Applies the properties of a selected object to other objects.

PROPERTIES

Controls properties of existing objects.

System Variables

HPANNOTATIVE

Controls whether a new hatch pattern is annotative.

HPASSOC

Controls whether hatches and fills are associative.

HPBACKGROUNDCOLOR

Controls the background color for hatch patterns.

HPCOLOR

Sets a default color for new hatches.

HPDRAWORDER

Controls the draw order of hatches and fills.

HPINHERIT

Controls whether to inherit the hatch origin when using the Inherit Properties option in HATCH and HATCHEDIT.

HPLAYER

Specifies a default layer for new hatches and fills.

HPSCALE

Sets the hatch pattern scale factor.

HPSEPARATE

Controls whether a single hatch object or separate hatch objects are created when operating on several closed boundaries.

HPTRANSPARENCY

Sets the default transparency for new hatches and fills.

MIRRHATCH

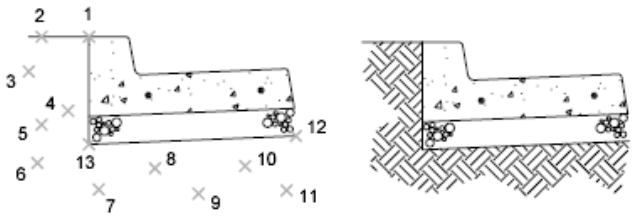
Controls how MIRROR reflects hatch patterns.

Control the Display of Hatch Boundaries

Hide or remove boundary objects to create hatches without borders.

To create hatches that have no boundary objects, do one of the following:

- Erase the boundary objects of an existing hatch.
- Trim an existing hatch to objects that cross the edges of the hatch. After trimming, erase the objects.
- Define hatch boundary points with the Draw option of the -HATCH command.



To hide a hatch's boundary objects, assign the boundary objects to a different layer than the hatch object, and then turn off or freeze the layer of the boundary objects. This method maintains hatch associativity.

See also:

- Reshape a Hatch or Fill on page 1392

To create an unbounded hatch

1. At the Command prompt, enter **-hatch**.
2. Enter **p** to specify Properties.
3. Enter the name of the pattern. For example, enter **earth** to specify the EARTH pattern.
4. Specify the scale and angle for the hatch pattern.
5. Enter **w** to specify Draw Boundary.
6. Enter **n** to discard the polyline boundary once the hatch area has been defined.
7. Specify points to define the boundary. Enter **c** to close the polyline boundary.
8. Press Enter twice to create the hatch.

Quick Reference

Commands

-HATCH

Uses command prompts to fill an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

PROPERTIES

Controls properties of existing objects.

System Variables

HPANG

Sets the angle for new hatch patterns.

HPBOUNDRETAIN

Controls whether boundary objects are created for new hatches and fills.

HPCOLOR

Sets a default color for new hatches.

HPDLGMODE

Controls the display of the Hatch and Gradient dialog box and the Hatch Edit dialog box.

HPDOUBLE

Specifies hatch pattern doubling for user-defined patterns.

HPMAXAREAS

Sets the maximum number of enclosed areas that a single hatch object can have and still automatically switch between solid and pattern hatches during zoom operations.

HPMAXLINES

Sets the maximum number of hatch lines that are generated in a hatch operation.

HPNAME

Sets the default hatch pattern name.

HPOBJWARNING

Sets the number of hatch boundary objects that can be selected before displaying a warning message.

HPINHERIT

Controls whether to inherit the hatch origin when using the Inherit Properties option in HATCH and HATCHEDIT.

HPORIGIN

Sets the hatch origin point for new hatch patterns relative to the current user coordinate system.

HPORIGINMODE

Controls how the default hatch origin point is determined.

HPSCALE

Sets the hatch pattern scale factor.

HPSPACE

Sets the hatch pattern line spacing for user-defined patterns.

HPTRANSPARENCY

Sets the default transparency for new hatches and fills.

Control the Draw Order of Hatches and Fills

Specify the draw order for a hatch object to control whether it is displayed behind or in front of the hatch boundary, or behind or in front of all other objects.

This behavior is controlled by the HPDRAWORDER system variable.

In drawings that contain many hatch objects, use the HATCHTOBACK command to display all hatch objects behind all other objects.

Quick Reference

Commands

HATCHTOBACK

Sets the draw order for all hatches in the drawing to be behind all other objects.

System Variables

HPDRAWORDER

Controls the draw order of hatches and fills.

Modify Hatches and Fills

Modify hatch properties and boundaries, or re-create the boundaries hatch objects.

Modify Hatch Properties

Modify the properties of hatch objects directly or copy them from another hatch object.

The following tools are available for modifying hatch properties:

- **Hatch panel controls.** Display on the ribbon by selecting a hatch object.
- **Hatch Edit dialog box.** Access the dialog box with HATCHEDIT.
- **Properties palette.**
- **Hatch shortcut menu.** Access the menu by right-clicking a hatch object.
- **Hatch dynamic menu.** Access the menu by hovering over the control grip on a selected hatch.
- **Command line.** Enter -HATCHEDIT.

Copy the properties of one hatch to another using the following methods:

- **Inherit Properties button in the Hatch Edit dialog box.** Copy all hatch-specific properties.
- **Match Properties command.** Use MATCHPROP to copy general properties and hatch-specific properties, with the exception of the hatch origin.

See also:

- Control How Overlapping Objects Are Displayed on page 587

Quick Reference

Commands

HATCHEDIT

Modifies an existing hatch or fill.

HATCHTOBACK

Sets the draw order for all hatches in the drawing to be behind all other objects.

MATCHPROP

Applies the properties of a selected object to other objects.

PROPERTIES

Controls properties of existing objects.

System Variables

HPDLGMODE

Controls the display of the Hatch and Gradient dialog box and the Hatch Edit dialog box.

HPINHERIT

Controls whether to inherit the hatch origin when using the Inherit Properties option in HATCH and HATCHEDIT.

Modify Hatch Alignment, Scale, and Rotation

Shift, scale, or rotate hatch patterns to align them with existing objects.

To shift a hatch pattern, relocate the origin point of the hatch object. The same tools in the user interface as listed in Modify Hatch Properties on page 1389 include options for specifying a new origin point, specifying a different rotation angle, and changing the scale of the hatch pattern.

In some cases, it might be easier to move or rotate the user coordinate system to align with existing objects, and then recreate the hatch.

To modify the properties of a hatch object

1. Select a hatch object.
2. Change the desired hatch settings from the Hatch Editor tab on the ribbon, the Modify menu, Modify toolbar, or enter **hatchedit** at the Command prompt.
3. In the user interface, a panel in the Hatch Editor tab, or the Hatch Edit dialog box, change the hatch settings or properties as needed.

Quick Reference

Commands

HATCHEDIT

Modifies an existing hatch or fill.

MATCHPROP

Applies the properties of a selected object to other objects.

PROPERTIES

Controls properties of existing objects.

System Variables

HPDLGMODE

Controls the display of the Hatch and Gradient dialog box and the Hatch Edit dialog box.

HPINHERIT

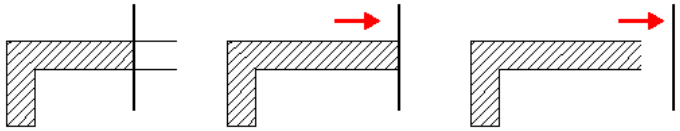
Controls whether to inherit the hatch origin when using the Inherit Properties option in HATCH and HATCHEDIT.

Reshape a Hatch or Fill

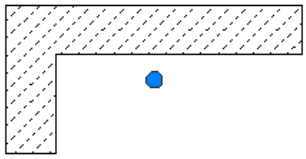
Reshape an associative hatch by modifying the boundary objects. Reshape a nonassociative hatch by modifying the hatch object.

Modify the Extents of Associative Hatches and Fills

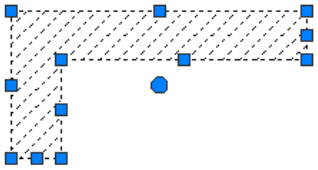
If you modify the boundary objects of an associative hatch, and the result maintains a closed boundary, the associated hatch object is automatically updated. If the changes result in an open boundary, the hatch loses its associativity with the boundary objects, and the hatch remains unchanged.



When you select an associative hatch object, it displays a circular grip, called the *control grip*, at the center of the hatch extents. Hover over the control grip to display a shortcut menu with several hatch options, or right-click to display additional options.



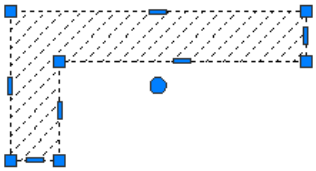
You can also change the hatch object by editing the grips of the associated boundary objects. To easily select all of the objects in a complex boundary, use the Display Boundary Objects option.



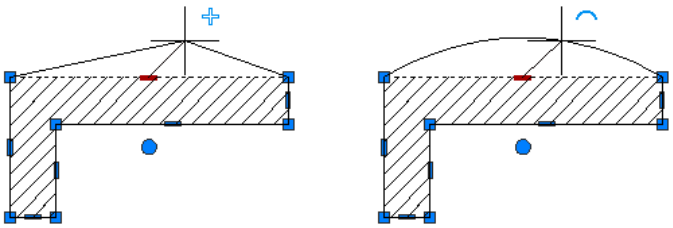
If the boundary object is a polyline or spline, *multi-functional grips* are displayed. For more information, see Use Object Grips on page 778.

Modify the Extents of Non-associative Hatches and Fills

When you select a non-associative hatch, multi-functional grips are displayed on the hatch. Use these grips to modify the hatch extents and some several hatch properties.



When you *hover* over a grip on a nonassociative hatch object, a grip menu displays several edit options based on the type of grip. For example, a linear segment grip has an option to convert the segment to an arc, or to add a vertex.



NOTE For drastic changes, you can use TRIM to reduce the area covered by a hatch object, or EXPLODE to disassemble a hatch into its component objects.

See also:

- Modify Objects Using Grips on page 777

Quick Reference

Commands

EXPLODE

Breaks a compound object into its component objects.

TRIM

Trims objects to meet the edges of other objects.

System Variables

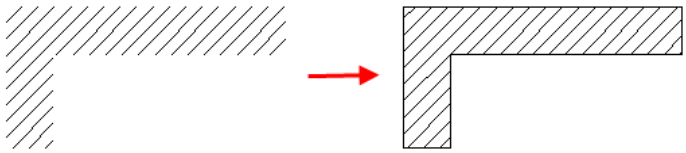
GRIPS

Controls the display of grips on selected objects.

Re-create the Boundary of a Hatch or Fill

Create a new boundary object for a non-associative or an unbounded hatch or fill.

Use the Recreate Boundary option to generate a closed polyline or a region object around a selected hatch or fill. You can also specify that the new boundary object is associated with the hatch.



To re-create the boundary object of a hatch or fill (ribbon turned on)

1. Select the hatch object.
2. On the ribbon, Boundaries panel, click Recreate Boundary.
3. At the prompt, specify the type of object to create as the new boundary, and whether to associate the boundary with the hatch.
4. Press Enter, or click Close on the ribbon to apply the hatch and exit the command.

To re-create the boundary object of a hatch or fill (ribbon turned off)

1. Start the Modify Hatch command from a menu or toolbar, or enter **hatchedit** at the Command prompt. 
2. Select the hatch object.
3. In the Hatch Edit dialog box, under Boundaries, click Recreate Boundary.
4. Specify the type of object to create as the new boundary.
5. Specify whether to associate the boundary with the hatch object.
6. Click OK to apply the hatch.

Quick Reference

Commands

HATCHEDIT

Modifies an existing hatch or fill.

System Variables

HPDLGMODE

Controls the display of the Hatch and Gradient dialog box and the Hatch Edit dialog box.

HPBOUND

Controls the object type created by HATCH and BOUNDARY.

HPBOUNDRETAIN

Controls whether boundary objects are created for new hatches and fills.

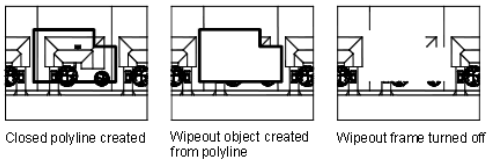
HPSEPARATE

Controls whether a single hatch object or separate hatch objects are created when operating on several closed boundaries.

Create a Blank Area to Cover Objects

Create a polygonal area, called a *wipeout* to mask underlying objects with the current background color.

A *wipeout object* covers existing objects with a blank area to make room for notes or to mask details. This area is defined by the wipeout frame, which you can turn on for editing, and turn off for plotting.



Use the WIPEOUT command both for creating a wipeout object, and for controlling whether wipeout frames are displayed or hidden in the drawing.

If a polyline is used to create a wipeout object, the polyline must be closed, contain line segments only, and have zero width.

Use Wipeout Objects on a Layout

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.

NOTE 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 587

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 the command.

To turn all wipeout frames on or off

1. Click Annotate tab ➤ Markup panel ➤ Wipeout. 
2. At the prompt, enter **f** (Frames).
3. Enter **on** or **off**, and press Enter.

Quick Reference

Commands

WIPEOUT

Creates a wipeout object, and controls whether wipeout frames are displayed in the drawing.

Notes and Labels 29

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.

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.

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.

Create Text

You can create text using several methods, depending on your needs.

See also:

- Use Fields in Text on page 1458

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 annotative on page 2289 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 1335.

See also:

- Scale Annotations on page 1324
- Create Annotative Text on page 1335

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.

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.

Quick Reference

Commands

MTEXT

Creates a multiline text object.

TEXT

Creates a single-line text object.

System Variables

MTEXTED

Sets the application for editing multiline text objects.

STYLE

Creates, modifies, or specifies text styles.

TEXTED

Specifies the user interface displayed for editing single-line text.

TEXTFILL

Controls the filling of TrueType fonts while plotting.

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.

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** to type text at the Command prompt instead of in-place.

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.

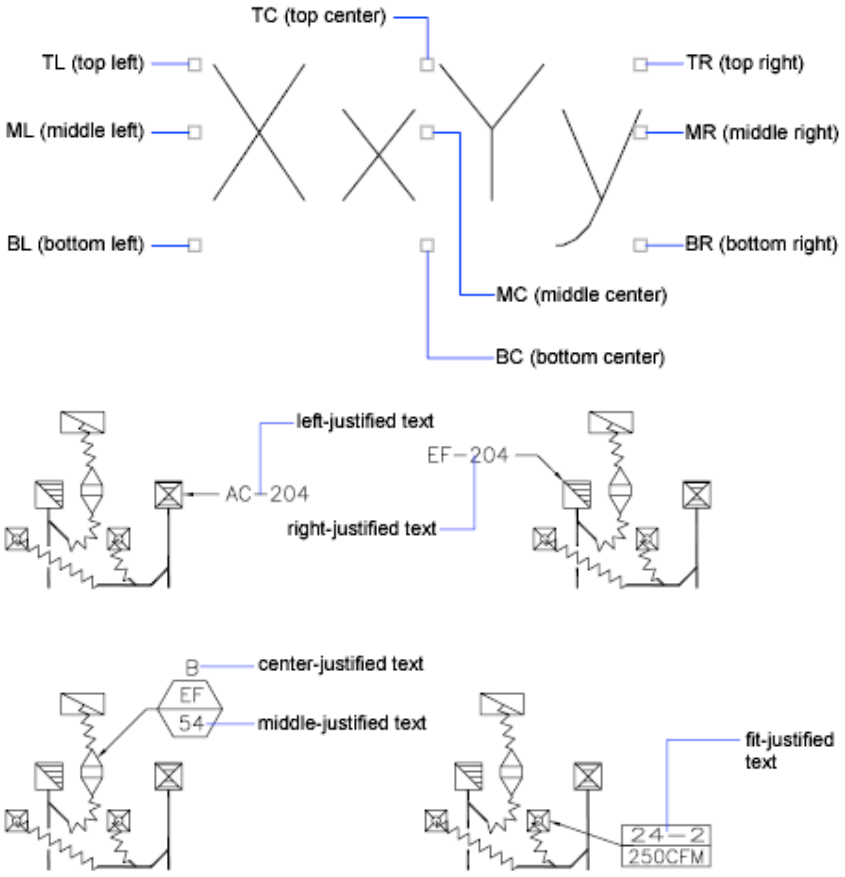
The text styles used for single-line text are the same as those used for multiline text. When you create text, you assign an existing style by entering its name at the Style prompt. If you need to apply formatting to individual words and characters, use multiline text instead of single-line text.

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 TEXTED 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 1458

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.

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.

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.

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

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 MIRROR reflects text.

QTEXTMODE

Controls how text is displayed.

TEXTED

Specifies the user interface displayed for editing single-line text.

TEXTEVAL

Controls how text strings entered with TEXT (using AutoLISP) or with -TEXT are evaluated.

TEXTFILL

Controls the filling of TrueType fonts while plotting.

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.

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 In-Place Text Editor. You can also type text at the Command prompt if you use -MTEXT. 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 In-Place Text Editor allows you to adjust the bounding box that defines the size of the multiline text object, as well as create and edit tabs and indents on the ruler along the top. The editor is transparent, as you create text, you can see whether the text overlaps other objects.

To turn off transparency while you work, right-click in the In-Place Text Editor and click Opaque Background from the shortcut 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.

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.

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.

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.

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.

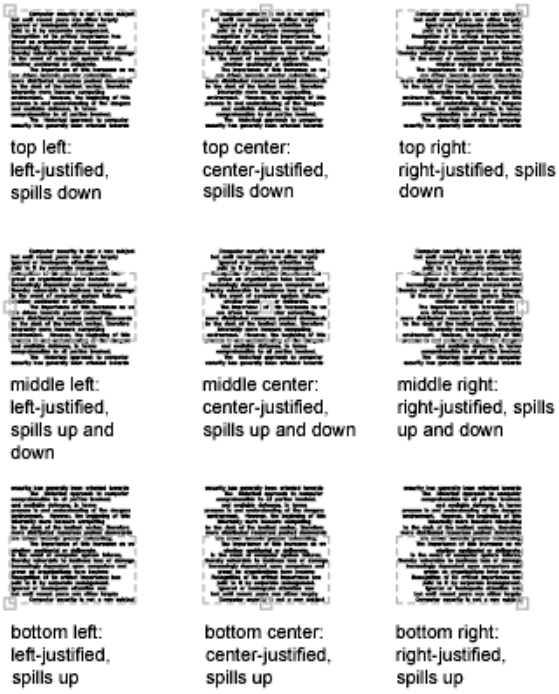
Justify Multiline Text

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.

Quick Reference

Commands

PROPERTIES

Controls properties of existing objects.

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 In-Place Text Editor shortcut menu resets the character attributes of selected text to the current text style and text color.

The text height setting specifies the height of capitalized text. For more information about how height is calculated, see MTEXT.

See also:

- Work with Text Styles on page 1469
- Use an Alternate Text Editor on page 1497

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.

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.

TEXTSTYLE

Sets the name of the current text style.

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 ➤ 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 a word processor and paste the list into a multiline text, the bullets that are displayed as empty circles might not be formatted like other bullets in multiline text. This is because the bullet might be a letter, such as *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.

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.

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.

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 (}).

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.

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.

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.

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.

TEXTQLTY

Sets the resolution tessellation fineness of text outlines.

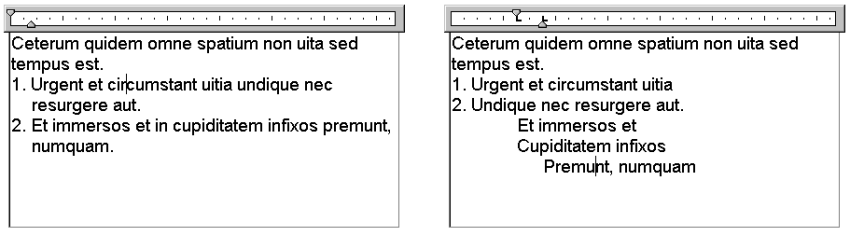
Indent Multiline Text and Use Tabs

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.

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.

TEXTQLTY

Sets the resolution tessellation fineness of text outlines.

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.

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.

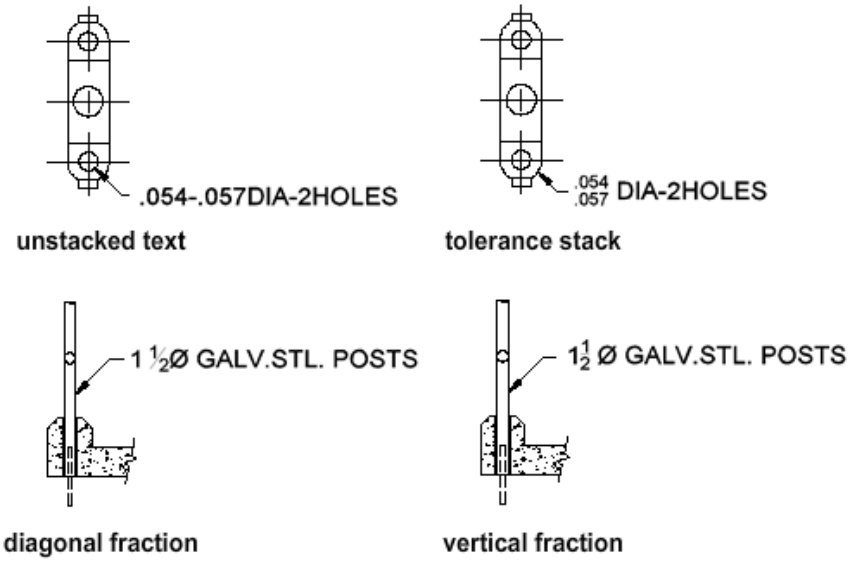
TSPACETYPE

Controls the type of line spacing used in multiline text.

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 right-click. From the shortcut menu, click Stack.

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 Stack Properties 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:

- Work with Text Styles on page 1469
- Use an Alternate Text Editor on page 1497

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.

To change stack properties

1. Double-click the multiline text object you want to edit.
2. In either the MTEXT ribbon contextual tab (Formatting tab drop-down) 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.

To unstack text

1. Double-click the multiline text object you want to edit.
2. In either the MTEXT ribbon contextual tab (Formatting tab drop-down) 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.

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.

Create and Edit Columns in Multiline Text

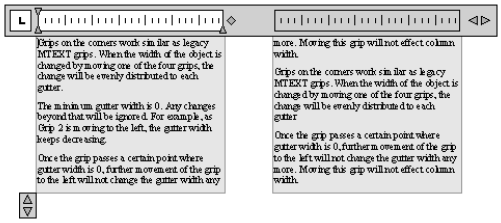
You can create and edit multiple columns using the In-Place Text Editor column options and column grips.

Multiple columns can be created and edited with the In-Place Text Editor and through grip editing. 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 in the In-Place Text Editor, the columns will be in a frame. The ruler bar spans across all columns, but is only active for the current column.



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 Properties Palette

You will be able to select Static or Dynamic columns, turn off columns and change column and gutter width through the Properties 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.

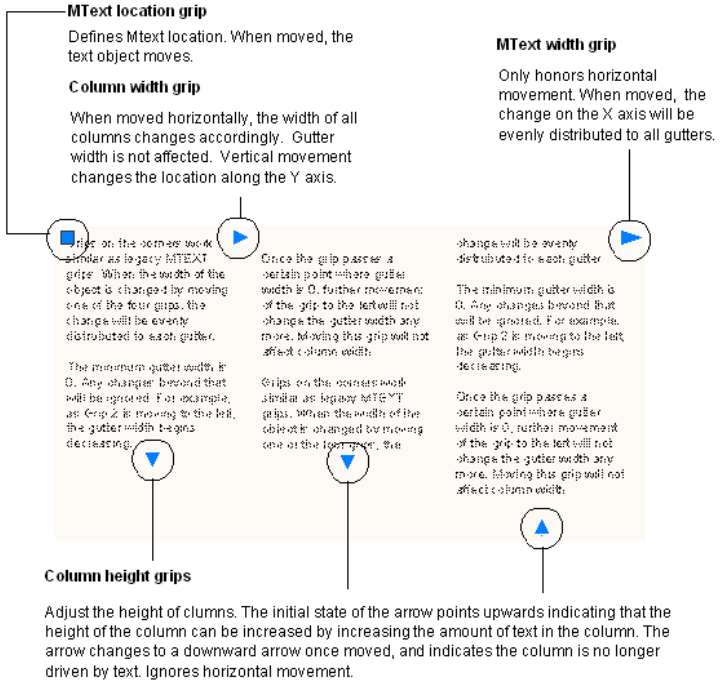
   NOTE The arrows on the ruler on the upper right side only adjust gutter width, not column width.

To adjust columns using grips

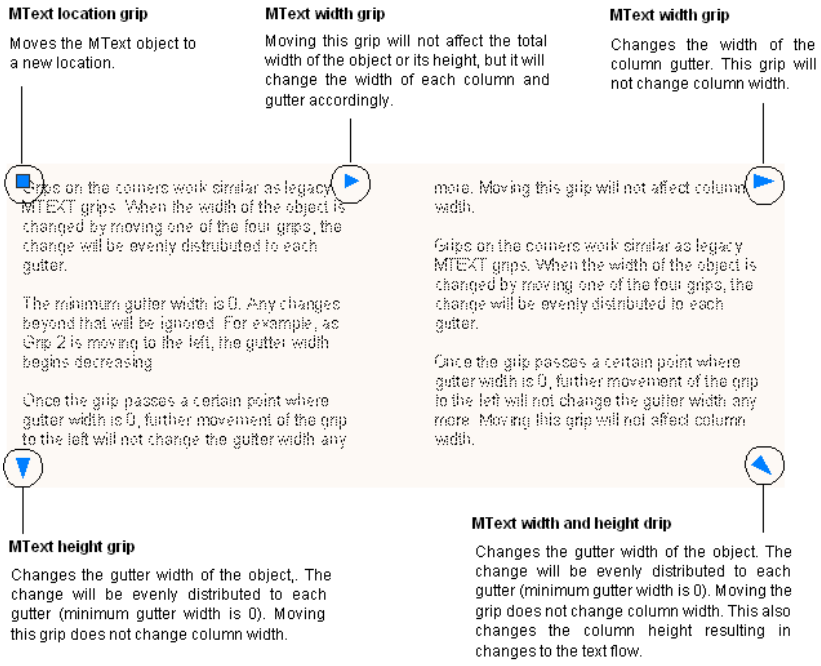
1. Select an area outside the mtext object. The In-Place Text Editor toolbar will disappear.
2. Click once in the text area and grips will appear.

   Grips control the location of the mtext object, the gutter width, and vertical and horizontal movement of colmns.

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.

Import Text from External Files

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 on page 1826

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.

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.

Create Leaders

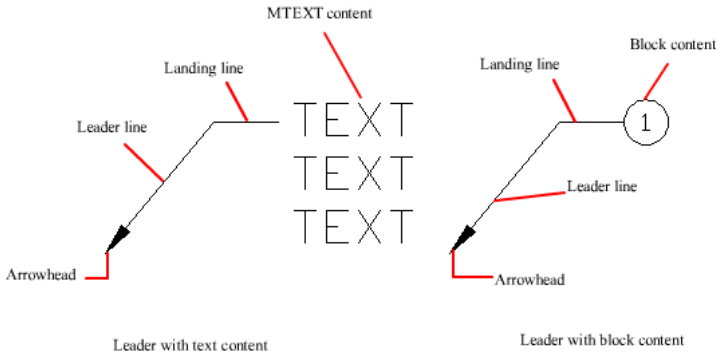
You can create, modify and add content to a leader object.

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.

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.

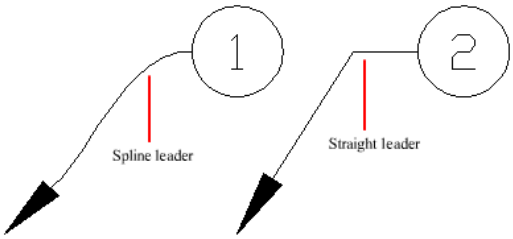
MLEADERSCALE

Sets the overall scale factor applied to multileader objects.

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

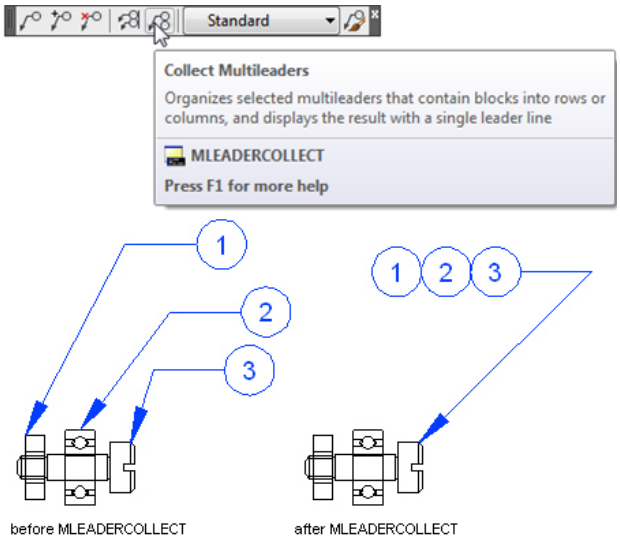
Multileader objects can contain multiple leader lines, each of which can have one or more segments, so that one note can point to multiple objects in your drawing. You can modify the properties of leader segment in the Properties palette. Add leaders to or remove leaders from a multileader object with MLEADEREDIT or choose options from the leader grip menus (see Modify Leaders Using Grips on page 1449).

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 1344.

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 MLEADERCOLLECT, 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 MLEADERALIGN, selected multileaders can be aligned and evenly spaced as specified.

Associate Leaders with Objects

When associative dimensioning is turned on (DIMASSOC system variable), 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 1344
- Modify Leaders Using Grips on page 1449

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.

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
- Apply the new multileader style to an existing multileader object

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.

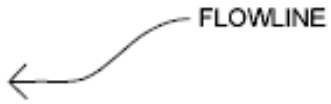
5 Enter **p** to specify a spline leader.

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.



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.

To create multiple leaders from the same annotation

1 Select the multileader.

2 Do one of the following:

- Hover over the landing grip, and choose Add Leader from the grip menu.
- Click Annotate tab ➤ Multileaders panel ➤ Add Leader. 

3 Specify the endpoint for the new leader.

To remove leaders from an annotation

1. Select the multileader.
2. Do one of the following:
   - Hover over a leader endpoint grip, and choose Remove Leader from the grip menu.
   - On the Multileader toolbar, click Remove Leader, and then select the leaders you want to remove. Press ENTER.

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.

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.

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.

To change the properties of a multileader object

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.

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.

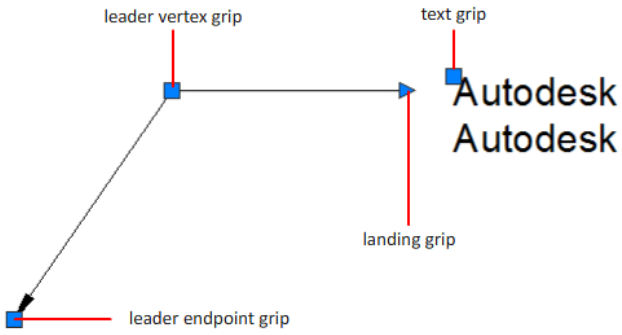
MLEADERSCALE

Sets the overall scale factor applied to multileader objects.

Modify Leaders Using Grips

You can make many leader edits directly using multi-functional grips. You can add and remove leaders, add and remove vertices, lengthen or move the landing line, or move the leader text.

Hover over a grip to access the option you want.



- From the Landing grip, you can choose:
  - *Stretch* to move the leader landing.
  - *Lengthen Landing* to extend the Landing line.
  - *Add Leader* to add one or more leader lines.
- From a leader endpoint grip, you can choose:
  - *Stretch* to move the leader endpoint.
  - *Add Vertex* to add a vertex to the leader line.
  - *Remove Leader* to delete the selected leader line.
- From a leader vertex grip, you can choose:
  - *Stretch* to move the vertex.
  - *Add Vertex* to add a vertex on the leader line.
  - *Remove Vertex* to delete the vertex.

See also:

- Modify Leaders Using Grips on page 1449

To modify leaders using grips

1. Select the multileader you want to modify.
2. Hover over a leader endpoint grip, a vertex grip, or the landing grip.
3. Click the option you want.
4. Click to specify a location for the action you're performing.

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.

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.

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.

Quick Reference

Commands

MLEADERSTYLE

Creates and modifies multileader styles.

System Variables

CMLEADERSTYLE

Sets the name of the current multileader style.

Add Content to a Leader

Leaders can contain multiline text or blocks to label parts of your drawing.

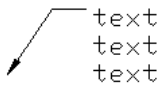
Leaders Containing Multiline Text

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.

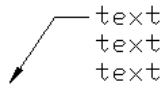
You can create annotative multileaders with text as content. The text content will be scaled according to the specified scale representation. Width, justification, attachment, and rotation settings for text content can be different depending on the specified scale representation. Actual text content cannot change with the scale representation.

There are several options for placing multiline text as content in a leader object.

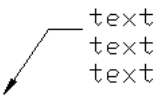
Top of top line



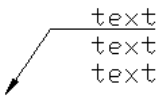
Middle of top line



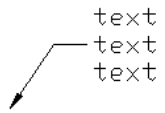
Bottom of top line



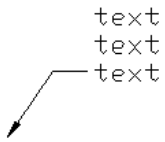
Underline top line



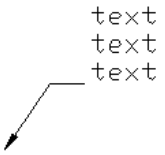
Middle of text



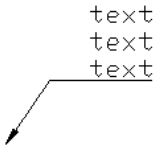
Middle of bottom line



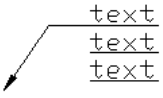
Bottom of bottom line



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 1458

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.

TEXTQLTY

Sets the resolution tessellation fineness of text outlines.

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.

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.

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 releases, 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.

See also:

- Work with AutoCAD Drawings in AutoCAD LT on page 1963

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.

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.

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.

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.

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 options 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 annotative on page 2289 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 1335.

See also:

- Scale Annotations on page 1324
- Create Annotative Text on page 1335

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.

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.

Assign Text Fonts

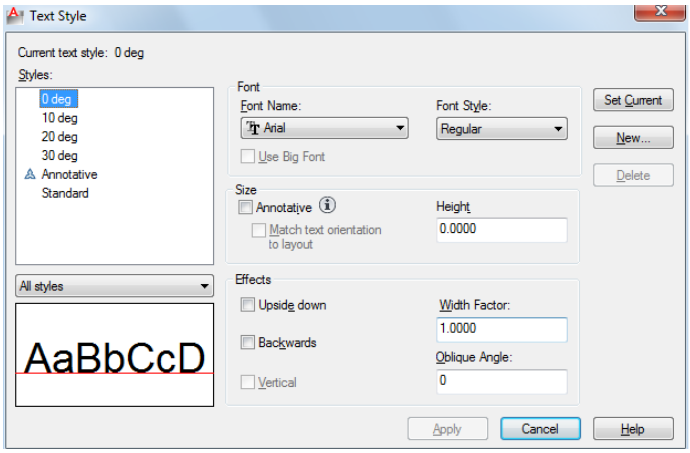
You can assign a text font as part of the text style definition. Several factors depend on the type of text you are working with.

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.

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 the operating system. Because SHX fonts are not recognized by the operating system, 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 1479

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.

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.

TEXTQLTY

Sets the resolution tessellation fineness of text outlines.

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 Asian sets, or more commonly known as 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 1477

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.

Quick Reference

Commands

STYLE

Creates, modifies, or specifies text styles.

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 (or Asian Set), 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 1474.

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 TEXT 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*. 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.

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.

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.

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.

Quick Reference

Commands

STYLE

Creates, modifies, or specifies text styles.

System Variables

TEXTSIZE

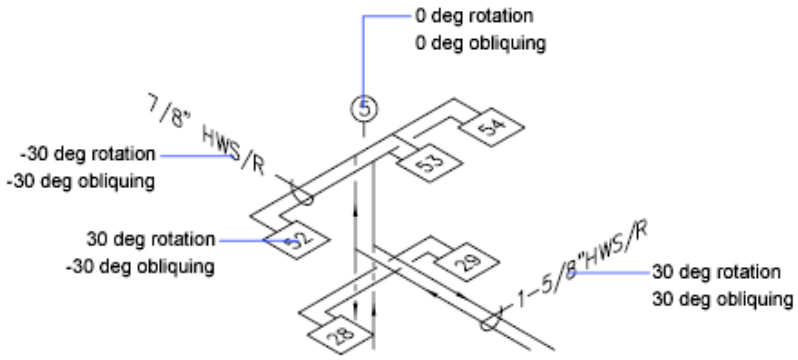
Sets the default height for new text objects drawn with the current text style.

TEXTSTYLE

Sets the name of the current text style.

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.

Quick Reference

Commands

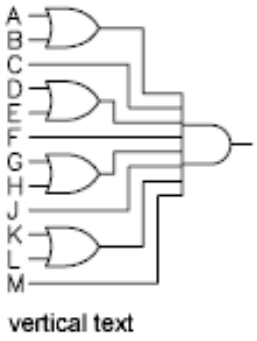
STYLE

Creates, modifies, or specifies text styles.

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.



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 releases, 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.

Quick Reference

Commands

STYLE

Creates, modifies, or specifies text styles.

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.

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 MIRROR reflects text.

MTEXTTOOLBAR

Controls the display of the Text Formatting toolbar.

TEXTFILL

Controls the filling of TrueType fonts while plotting.

TEXTQLTY

Sets the resolution tessellation fineness of text outlines.

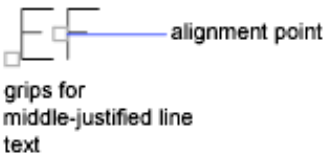
Change Single-Line Text

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.

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.

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

TEXTED

Specifies the user interface displayed for editing single-line text.

TEXTFILL

Controls the filling of TrueType fonts while plotting.

TEXTQLTY

Sets the resolution tessellation fineness of text outlines.

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 the following:

- Text style assignment
- Justification
- Width
- Rotation
- Line spacing

In addition, you can use the following to modify individual formatting, such as boldface and underlining, and width for multiline text objects:

- Text Editor ribbon contextual tab
- Text Formatting toolbar
- In-Place Text Editor
- Grips

Change Text Location

You can use many of the common modifying commands and grips to move multiline text objects. A multiline text object has grips at the four corners of the text boundary and, in some cases, at the justification point.

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 1469
- Control the Display of Polylines, Hatches, Gradient Fills, Lineweights, and Text on page 582

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.

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 MIRROR reflects text.

MTEXTED

Sets the application for editing multiline text objects.

MTEXTTOOLBAR

Controls the display of the Text Formatting toolbar.

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.

TEXTQLTY

Sets the resolution tessellation fineness of text outlines.

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
@ (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

Character	Definition
` (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.

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.

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 45.

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.

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.

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

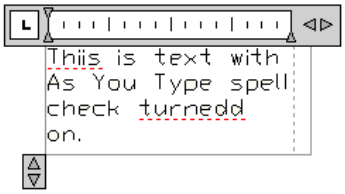
- Single and multiline text
- Dimension text
- Multileader text
- Text within block attributes
- Text within xrefs

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.

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.

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.

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.

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.

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.

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 AutoCAD
\~	Inserts a nonbreaking space	Autodesk AutoCAD\~LT	Autodesk AutoCAD LT
\\	Inserts a backslash	Autodesk \\AutoCAD	Autodesk \AutoCAD

Format codes for paragraphs

Format code	Purpose	Enter this …	To produce this …
\{...\}	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
\H*value*;	Changes to the text height specified in drawing units	Autodesk \H2;AutoCAD	Autodesk AutoCAD
\H*value*x;	Changes the text height to a multiple of the current text height	Autodesk \H3x;AutoCAD	Autodesk AutoCAD
\S...^...;	Stacks the subsequent text at the /, #, or ^ symbol	1.000\S+0.010^-0.000;	1.000 +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	A u t o d e s k
\Q*angle*;	Changes obliquing angle	\Q20;Autodesk	Autodesk
\W*value*;	Changes width factor to produce wide text	\W2;Autodesk	Autodesk

Format codes for paragraphs

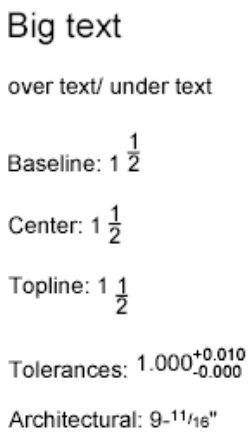
Format code	Purpose	Enter this …	To produce this …
\A	Sets the alignment value; valid values: 0, 1, 2 (bottom, center, top)	\A1;1\S1/2	
\P	Ends paragraph	Autodesk\PAutoCAD	Autodesk AutoCAD

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.

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.

Tables

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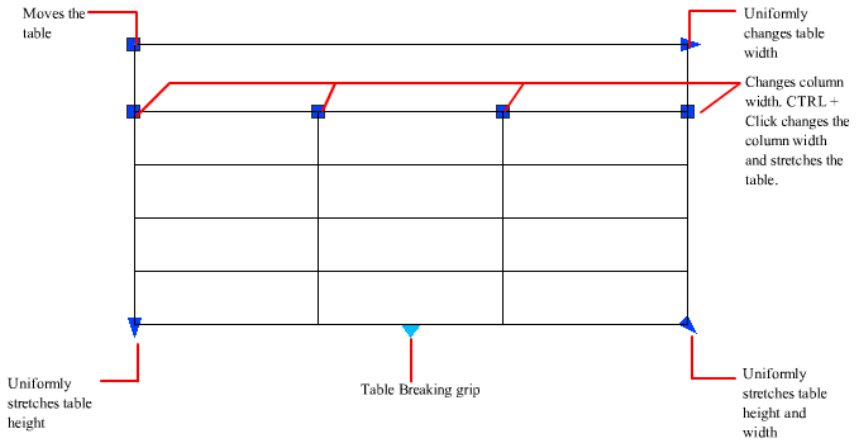
A table is a rectangular array of cells that contain annotation, primarily text but also blocks. Tables appear in many different forms on many of the sheets that make up drawing sets. In the AEC industry, tables are often referred to as “schedules” and contain information about the materials needed for 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.

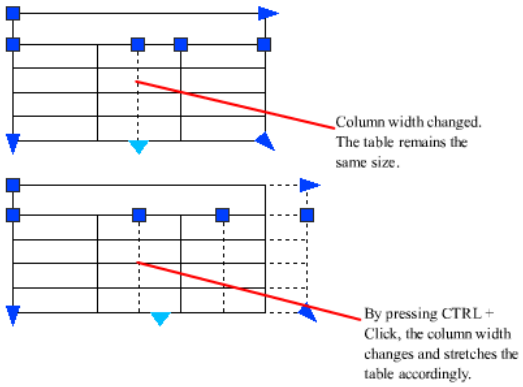
Create and Modify Tables

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 2317 or column on page 2294 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.

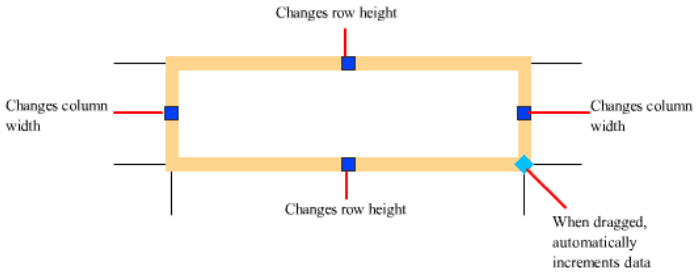


Break Tables into Multiple Parts

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, double-click to edit the cell text. You can also start entering text when a cell is highlighted to replace its current content.

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

- Insert and delete rows and columns
- Merge and unmerge cells
- Match cell styles
- 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 1514

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.

Quick Reference

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.

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.

Link a Table to External Data

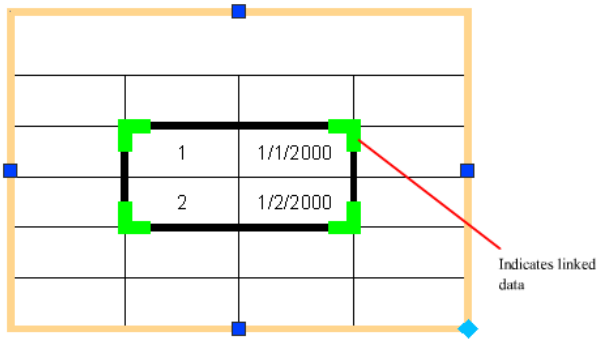
A table can be linked to data in a Microsoft Excel (**XLS**, **.XLSX**, or **CSV**) file. You can link to an entire spreadsheet, individual row, column, cell, or cell range in Excel.

NOTE Microsoft Excel must be installed to use Microsoft Excel data links. To link to the XLSX filetype, Microsoft Excel 2007 must be installed.

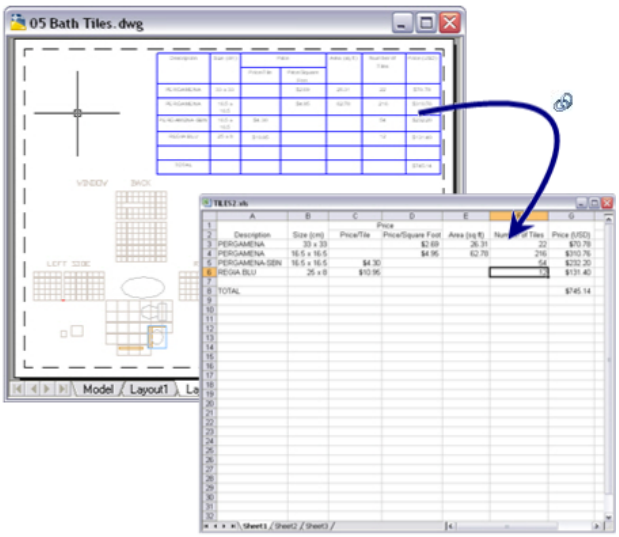
You can bring data from Microsoft Excel into a table in the following three ways:

- 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 on page 1969

To link to a table in an external spreadsheet

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, do one of the following:
   - Select Link to a Named Range. Click OK.

- 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.

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.

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 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

DATALINK

Displays the Data Link dialog box.

DATALINKUPDATE

Updates data to or from an established external data link.

DATALINKNOTIFY

Controls the notification for updated or missing data links.

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 1469

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. Click OK.

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.

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.

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.

Quick Reference

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.

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.

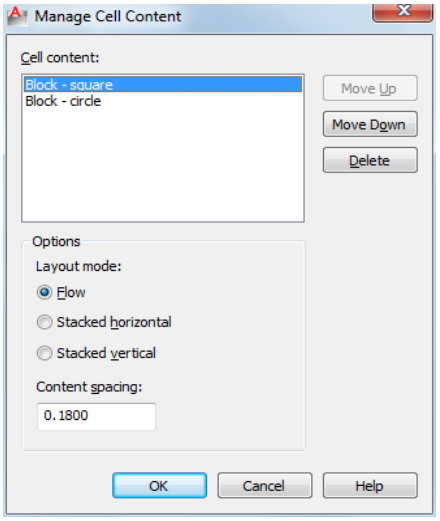
Add Text and Blocks to Tables

Table cell data can include text and multiple blocks.

When a table is created, the first cell is highlighted, 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 double-clicking in a selected cell or start entering text to replace the current content of a 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 1458
- Create Multiline Text on page 1410

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.

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.

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.

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.

To insert a block in a table cell

1. Select and right-click a cell. 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.

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.

Quick Reference

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.

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.

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 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.

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 1458

To add a formula to 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 one of the following:
   - Insert Formula ➤ Average
   - Insert Formula ➤ Sum
   - Insert Formula ➤ Count
   - Insert Formula ➤ Cell
3. Follow the prompts.
4. Edit the formula, if necessary.

5 To save your changes and exit the editor, click in the drawing outside the editor.

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, click in the drawing outside the editor.

The cell displays the result of the calculation.

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, do one of the following:
   - 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, click in the drawing outside the editor.

   The cell displays the result of the calculation.

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 an AutoFill option.
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

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.

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.

Dimensions and Tolerances 31

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.

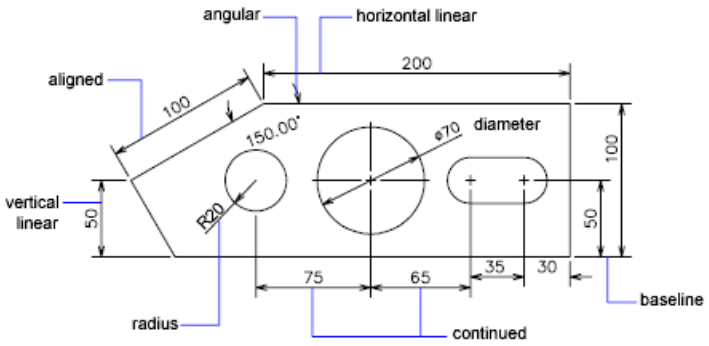
Overview of Dimensioning

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.

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

Creates a series of dimensions quickly from selected objects.

System Variables

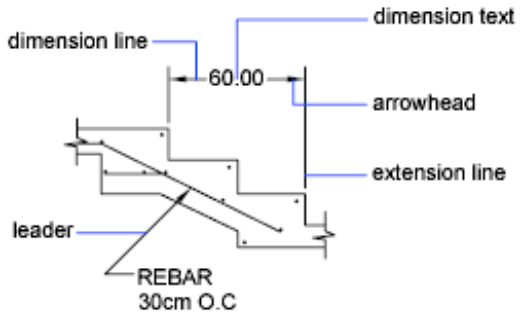
DIMASSOC

Controls the associativity of dimension objects and whether dimensions are exploded.

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.

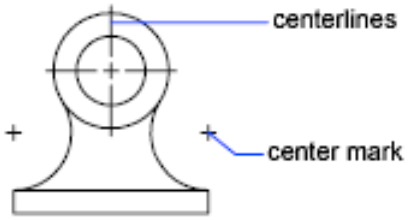
A *dimension line* indicates the direction and extent of a dimension. For angular dimensions, the dimension line is an arc.

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.

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.
- **Non-associative dimensions.** Selected and modified with the geometry they measure. Non-associative 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 non-associative 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.

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 non-associative elements of a dimension.

You can also use the Quick Select dialog box to filter the selection of associative or non-associative dimensions.

Special Situations and Limitations

You may need to use DIMREGEN to update associative dimensions after panning or zooming, 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, DGN, and PDF underlays

When selecting objects to dimension, make sure that the objects that you select do not 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.

For information about working with associative dimensions in combination with previous releases, see Save Drawings to Previous Drawing File Formats on page 1960.

See also:

- Change Dimension Associativity on page 1597

- Save Drawings to Previous Drawing File Formats on page 1960

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.

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.

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.

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.

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

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.

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.

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.

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.

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

Control Dimension Geometry

You can control the appearance of dimension lines, extension lines, arrowheads, and center marks.

Control Dimension Lines

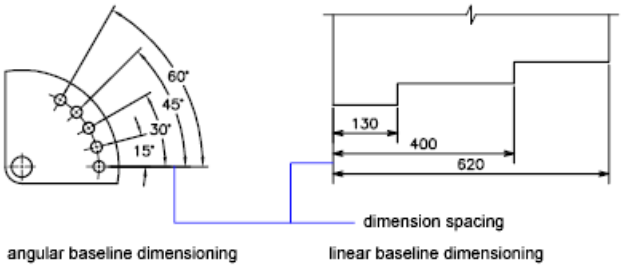
You can control dimension line properties including color, lineweight, and spacing.

You can control several aspects of a dimension line. You can

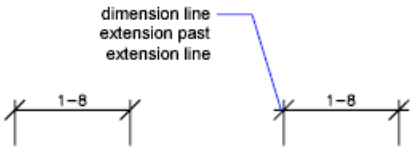
- Specify color and lineweight for visual effect and plotting
- Suppress the dimension line or, if the dimension line is broken by text, one or both halves



- 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.

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

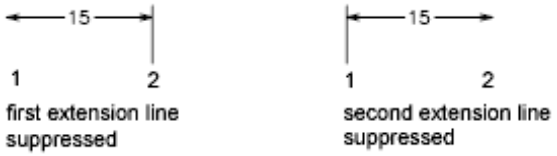
Controls whether a dimension line is drawn between the extension lines even when the text is placed outside.

Control Extension Lines

You can control extension line properties including color, lineweight, overshoot, and offset length.

You can

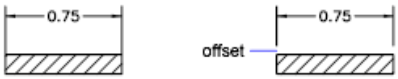
- Specify color and lineweight for visual effect and plotting
- Suppress one or both extension lines if they are unnecessary, or if there is not enough space



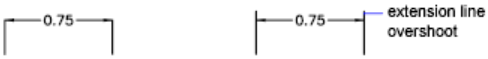
- Specify how far beyond from the dimension line the extension line extends (overshoot)



- Control the extension origin offset, the distance between the extension line origin, and the start of the extension line



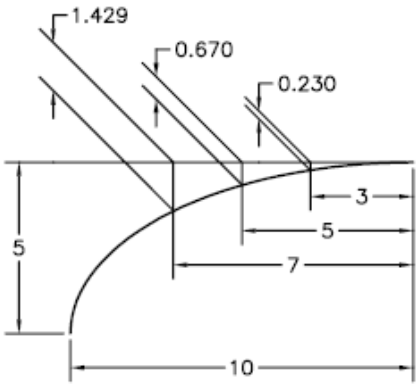
- Specify a fixed length for extension lines, as measured from the dimension line toward the extension line origin



- Specify a noncontinuous linetype, typically used for centerlines

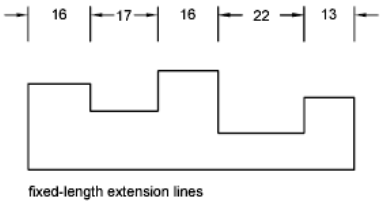


- Modify the angle of the extension lines of a selected dimension to make them oblique



Fixed-Length Extension Lines

You can specify a dimension style that sets the total length for extension lines starting from the dimension line toward the dimension origin point.



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 1578

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.

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

System Variables

DIMCLRE

Assigns colors to extension lines, center marks, and centerlines.

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.

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 1541

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.

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.

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 929.

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.

NOTE The insertion point a block is defined with affects its placement as a custom arrowhead on a dimension or leader. For information on changing the insertion point of a block, see Create Drawing Files for Use as Blocks on page 931.

If you use paper-space scaling, the scale factor is computed before applying it to the arrowhead size value.

See also:

- Create Blocks Within a Drawing on page 929
- Create Drawing Files for Use as Blocks on page 931

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.

Quick Reference

Commands

BLOCK

Creates a block definition from selected objects.

DIMSTYLE

Creates and modifies dimension styles.

WBLOCK

Saves selected objects or converts a block to a specified 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.

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.

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

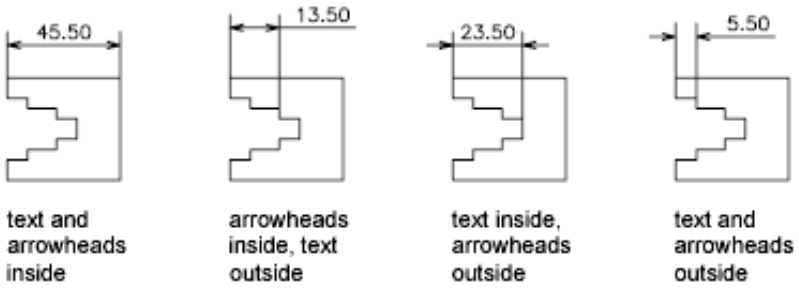
Dimension text and arrowheads usually appear between the extension lines when there is enough space. You can specify how these elements are placed when space is limited.

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 Modify/New Dimension Style dialog box, Fit tab.

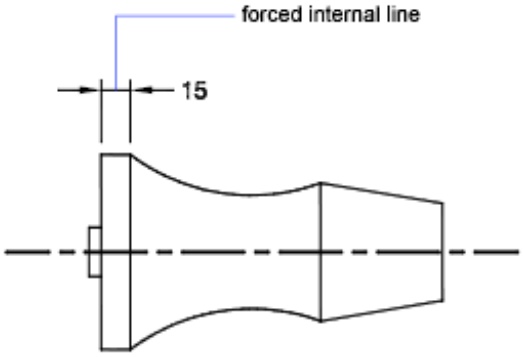
For example, you can specify that text and arrowheads be kept together. In this case, if there is not room for both between the extension lines, they are both placed outside. You can specify that if there is room for only text or arrowheads, then either text only or arrowheads only are placed between the extension lines.

The following illustrations show how the program applies a "best fit" for arrowheads and text.



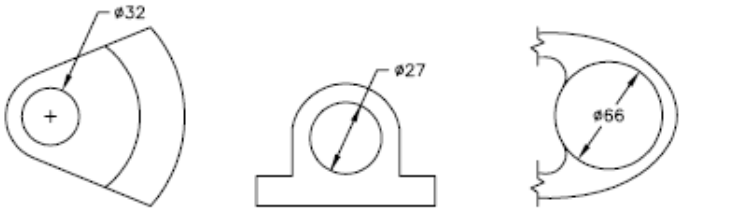
If there is no room for text between the extension lines, you can have a leader line created automatically. This is useful in cases where text outside the extension lines would interfere with other geometry, for example, in continued dimensions. Whether text is drawn to the right or the left of the leader is controlled by the horizontal justification setting in the Modify/New Dimension Style dialog box, Text tab. 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 Modify/New Dimension Style dialog box, Text tab, and whether you select the Draw Dim Line Between Ext Lines option on the Modify/New Dimension Style dialog box, Fit tab.



default horizontal placement-text outside circle, center mark, no forced interior line

user-defined horizontal placement-dimension line and arrows, no center mark, forced interior line

Text and Arrows inside with Inside Horizontal option selected

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.

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.

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.

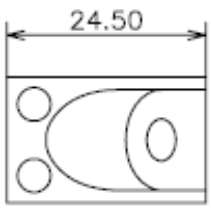
DIMUPT

Controls options for user-positioned text.

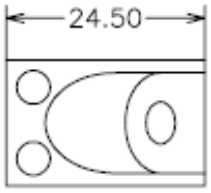
Control the Location of Dimension Text

You can locate dimension text manually and specify its alignment and orientation.

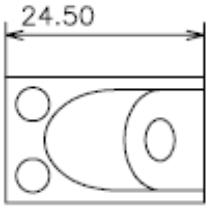
The program comes with several justification settings that facilitate compliance with international standards, or you can choose your own location for the text.



text centered above the dimension line



text centered horizontally and vertically

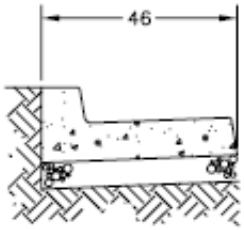


text left-justified above the dimension line

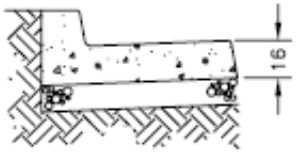
Many of the settings are interdependent. Example images in the Dimension Style Manager are updated dynamically to illustrate how text appears as you change the settings.

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

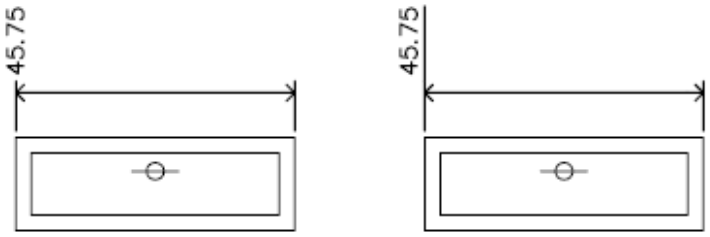


text outside extension lines aligned with dimension line

The default alignment is horizontal dimension text, even for vertical dimensions.

Position Dimension Text Horizontally

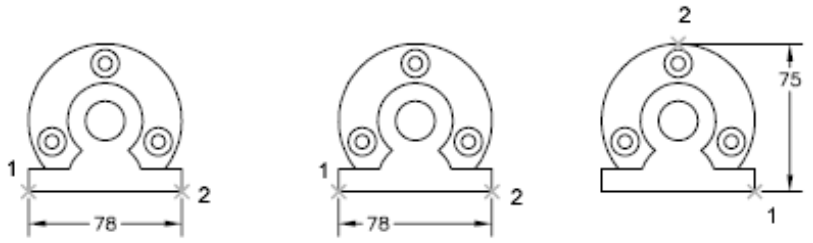
The position of the text along the dimension line in relation to the extension lines is referred to as text placement. To place text yourself when you create a dimension, use the Place Text Manually option on the Modify/New Dimension Style dialog box, Fit tab. Use the text placement options to automatically 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.

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.

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.

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.

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.

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 Modify/New Dimension Style dialog box, 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 1469.

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 in Offset from Dim Lim on the Modify/New Dimension Style dialog box, Text tab.

Example: User Text in Dimensions

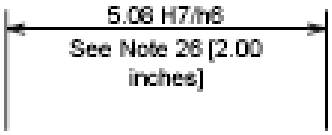
In this example, the primary dimension measurement is 5.08, and the alternate dimension measurement is 2.00. The primary units have the suffix *H7/h6*, and the alternate units have the suffix *inches*.

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.

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.

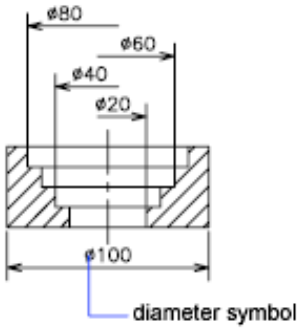
Control Dimension Values

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 (unicode 2205 and R, respectively).



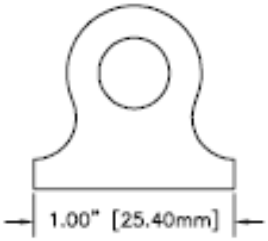
These settings are available on the Modify/New Dimension Style dialog box, Primary Units tab.

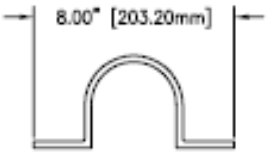
Control the Display of Alternate Units

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.

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.

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.

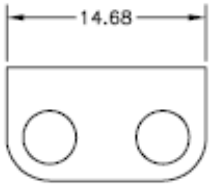
DIMTDEC

Sets the number of decimal places to display in tolerance values for the primary units in a dimension.

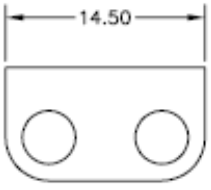
Round Off Dimension Values

You can round off the numeric values in dimensions and lateral tolerances.

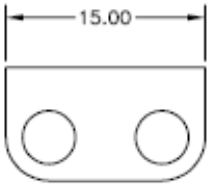
You can round off all dimension values except those for angular dimensions. For example, if you specify a round-off value of 0.25, all distances are rounded to the nearest 0.25 unit. The number of digits displayed after the decimal point depends on the precision set for primary and alternate units and lateral tolerance values.



round-off value set to default (0)



round-off value set to default .25



round-off value set to default 1

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.

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

System Variables

DIMRND

Rounds all dimensioning distances to the specified value.

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.

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.

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.

Display Lateral Tolerances

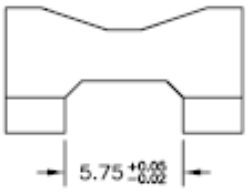
Lateral tolerances are values indicating the amount a measured distance can vary. You can control whether lateral tolerances are displayed and you can choose from several styles of lateral tolerances.

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.

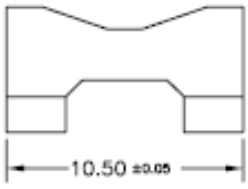
You can apply tolerances directly to a dimension by appending the tolerances to the dimension text. These dimension tolerances indicate the largest and smallest permissible size of the dimension. You can also apply geometric tolerances, which indicate deviations of form, profile, orientation, location, 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.

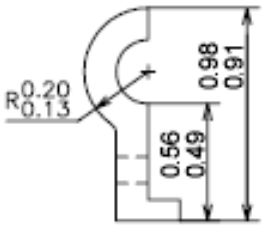


deviation tolerance



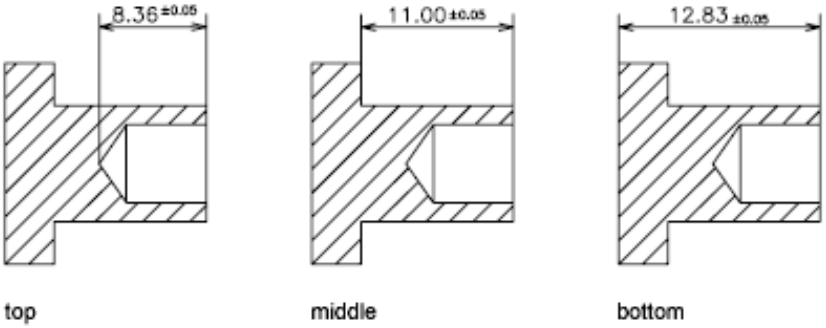
symmetrical deviation tolerance

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.

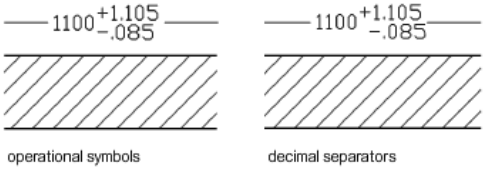


Format Lateral Tolerances

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 1621

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.

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.

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

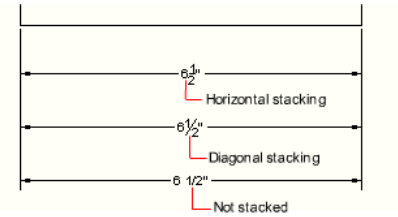
Controls the suppression of zeros in tolerance values.

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 Modify/New Dimension Style dialog box, Primary Units tab.

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.

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.

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 450
- Scale Annotations on page 1324

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.

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.

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.

Create Dimensions

You can create all of the standard types of dimensions.

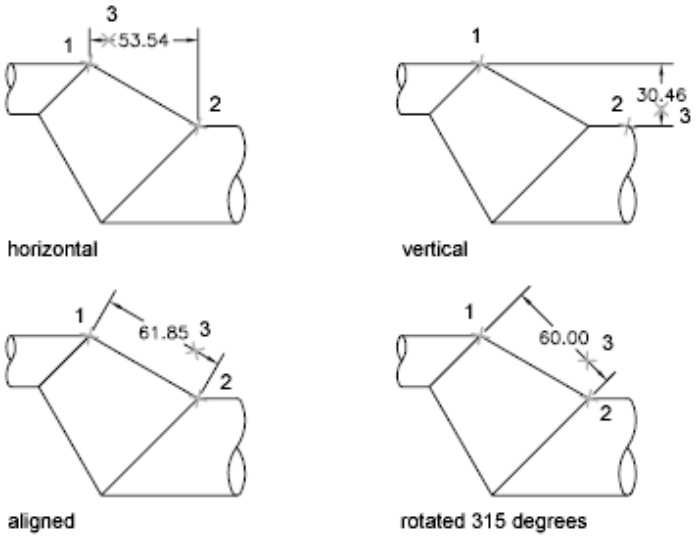
Create Linear Dimensions

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

Creates a series of dimensions quickly from selected objects.

System Variables

DIMDLI

Controls the spacing of the dimension lines in baseline dimensions.

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.

Quick Reference

Commands

DIMLINEAR

Creates a linear dimension.

System Variables

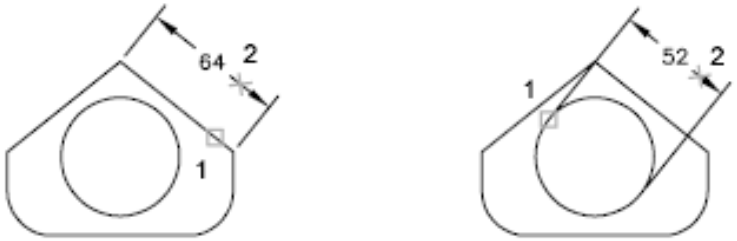
DIMEXO

Specifies how far extension lines are offset from origin points.

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.

Quick Reference

Commands

DIMALIGNED

Creates an aligned linear dimension.

DIMSTYLE

Creates and modifies dimension styles.

System Variables

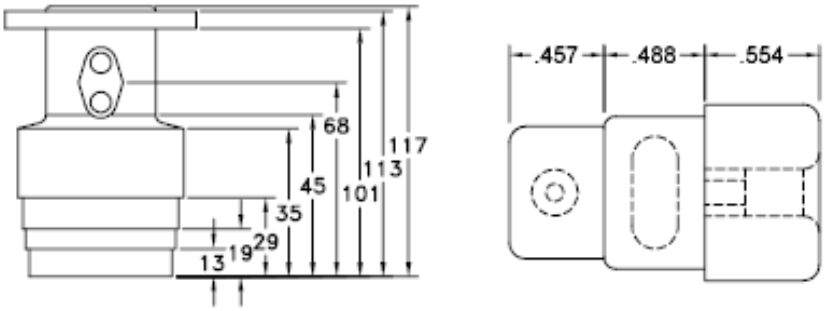
DIMEXO

Specifies how far extension lines are offset from origin points.

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.

TIP Hover over a dimension line endpoint grip to quickly access the Baseline or Continued commands from the grip menu. For more information, see Use Dimension Line Grips on page 1595.

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.

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.

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

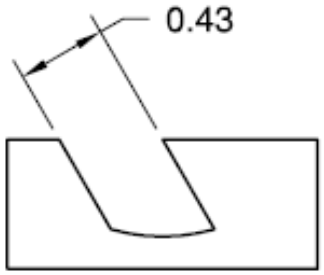
DIMDLI

Controls the spacing of the dimension lines in baseline dimensions.

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.

Quick Reference

Commands

DIMALIGNED

Creates an aligned linear dimension.

DIMLINEAR

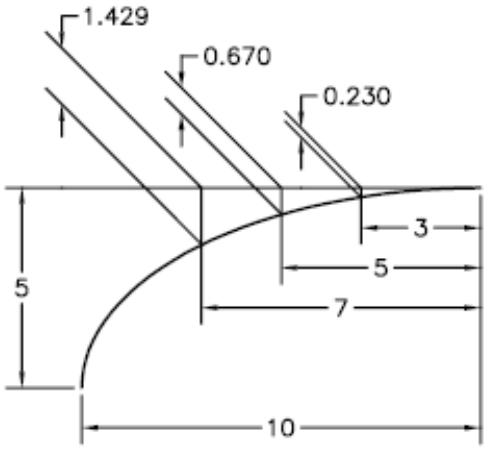
Creates a linear dimension.

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.

Quick Reference

Commands

DIMEDIT

Edits dimension text and extension lines.

Create Radial Dimensions

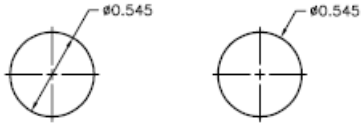
Radial dimensions measure the radii and diameters of arcs and circles with optional centerlines or a center mark.

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.



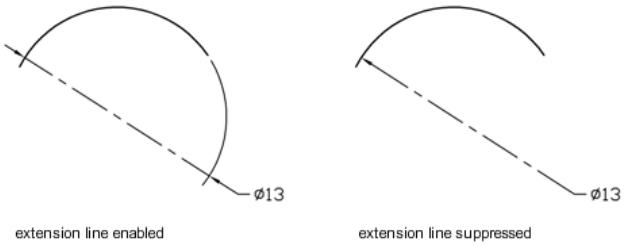
- DIMDIAMETER measures the diameter of an arc or circle, and displays the dimension text with the diameter symbol in front of it.



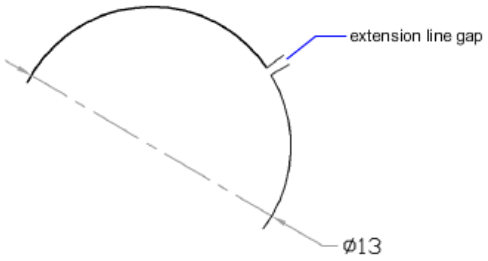
For horizontal dimension text, if the angle of the radial dimension line is greater than 15 degrees from horizontal, a hook line, also called a *dogleg* or *landing*, one arrowhead long, is created next to the dimension text.

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.



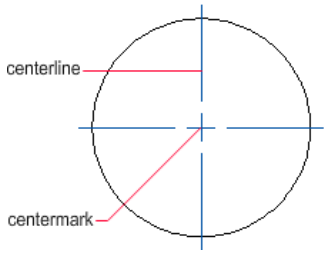
The DIMSE1 system variable controls whether or not a radial or diametric dimension will be drawn with an extension line when it is positioned off the end of an arc. When the display of the arc extension line is not suppressed, a gap between the arc and arc extension line is made. The size of the gap drawn is controlled with the DIMEXO system variable.



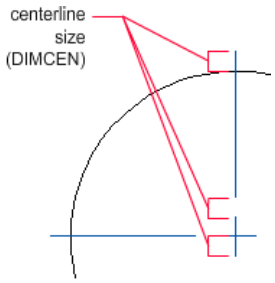
Control Centerlines and Center Marks

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 New/Modify Dimension Style dialog box, Symbols and Arrows tab, under Center Marks. You can also access this setting with the DIMCEN system variable.

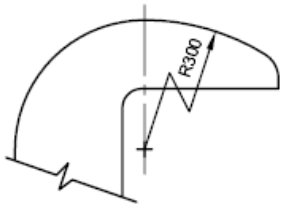


The size of the centerline is the length of the centerline segment that extends outside the circle or arc. It is also the size of the gap between the center mark and the start of the centerline. The size of the center mark is the distance from the center of the circle or arc to the end of the center mark.

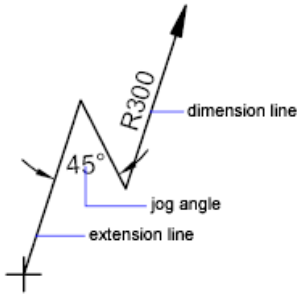


Create Jogged Radius Dimensions

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 New/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 1543

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.

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.

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.

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.

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.

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.

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

Creates a series of dimensions quickly 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.

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.

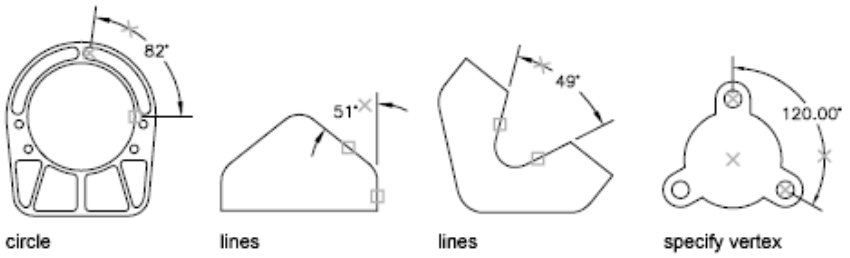
NOTE You can create baseline and continued angular dimensions relative to existing angular dimensions. Baseline and continued angular dimensions are limited to 180 degrees or less. To obtain baseline and continued angular dimensions larger than 180 degrees, use grip editing to stretch the location of 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

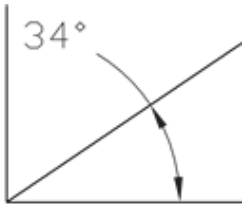
If you use an arc or a circle or three points to specify an angle, the program draws the dimension line arc between the extension lines. The extension lines are drawn from the angle endpoints to the intersection of the dimension line arc.



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.

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

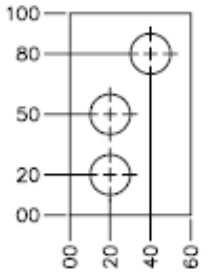
Sets the units format for angular dimensions.

DIMDEC

Sets the number of decimal places displayed for the primary units of a dimension.

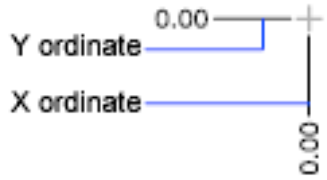
Create Ordinate Dimensions

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.



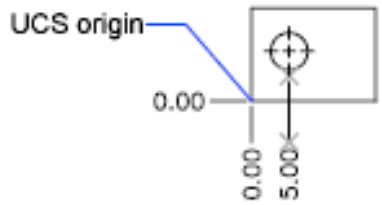
ordinate dimensions

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.



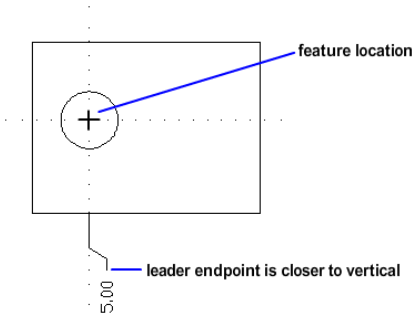
Locate the Datum

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.

Quick Reference

Commands

DIMORDINATE

Creates ordinate dimensions.

QDIM

Creates a series of dimensions quickly from selected objects.

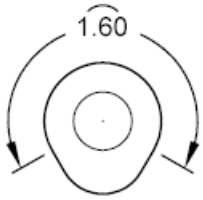
UCS

Sets the origin and orientation of the current UCS.

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. The placement style can be changed on the New/Modify Dimension Style dialog box, Symbols and Arrows tab.

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.

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.

Modify Existing Dimensions

You can modify all components of the existing dimension objects in a drawing either individually or by using dimension styles.

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.

Overview of Modifying Dimensions

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 reposition dimension text; and you can adjust the placement of linear dimensions so they are evenly spaced.

The easiest way to modify dimensions individually is to use the multi-functional dimension grips.

Modify Dimension Geometry

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

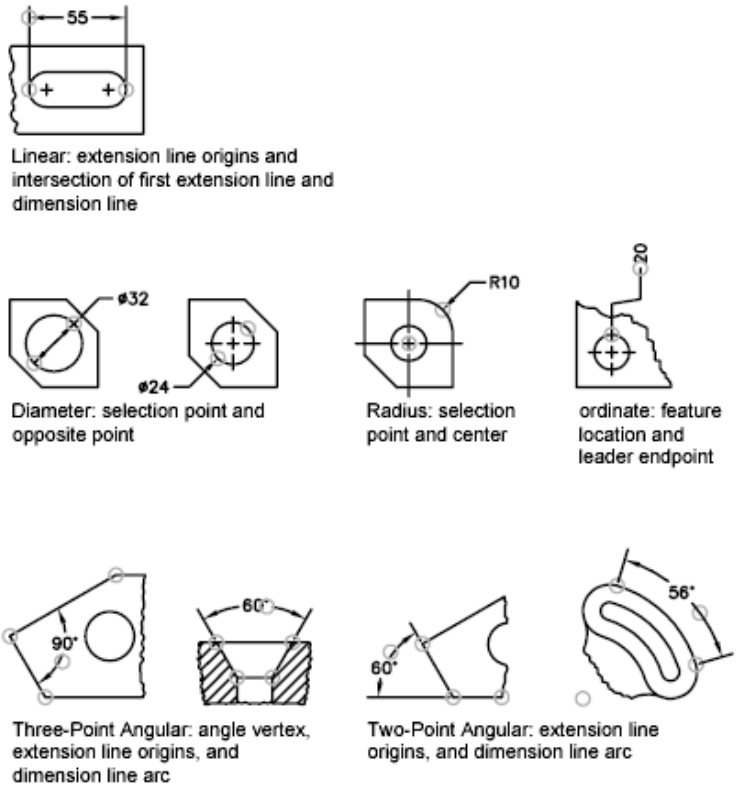
In other circumstances, a dimension may become partially associated. For example, if a linear dimension is associated with the endpoints of two geometric objects and one of the objects is erased, the remaining association is preserved. The disassociated end of the linear dimension may then be associated with another geometric object using DIMREASSOCIATE.

NOTE The Command prompt displays a warning message if a dimension is disassociated.

Modify Non-associative Dimensions

For non-associative 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.



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.

Use Dimension Line Grips

Hover over the grip on the endpoint of a dimension line to quickly access the following functionality:

- **Stretch.** Stretches the extension lines to move the dimension line farther away or closer to the object being dimensioned. Use command line prompts to specify a different base point or copy the dimension line. This is the default grip behavior.

- **Continue dimension.** Invokes the DIMCONTINUE command.
- **Baseline dimension.** Invokes the DIMBASELINE command.
- **Flip arrow.** Flips the direction of the dimension arrowhead.

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:

- Overview of Modifying Dimensions on page 1593
- Change Dimension Associativity on page 1597
- Control Dimension Geometry on page 1533

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.

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.

Change Non-associative Dimensions to Associative

You can change all the non-associative dimensions in a drawing to associative. Select all non-associative dimensions, and then use DIMREASSOCIATE to step through the dimensions, associating each one with locations on geometric objects.

Change Associative Dimensions to Non-associative

You can change all associative dimensions in a drawing to nonassociative dimensions. Select all associative dimensions, and then use DIMDISASSOCIATE to convert them into nonassociative dimensions.

See also:

- Associative Dimensions on page 1527
- Save Drawings to Previous Drawing File Formats on page 1960

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.

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

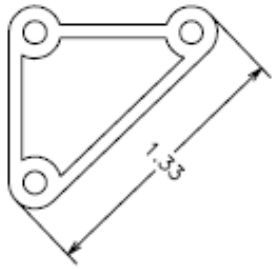
DIMASSOC

Controls the associativity of dimension objects and whether dimensions are exploded.

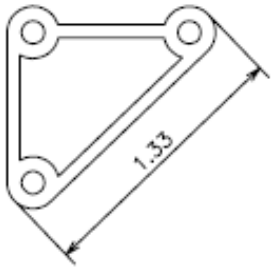
Modify Dimension Text

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.



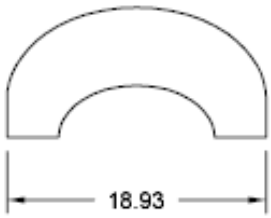
dimension text rotated



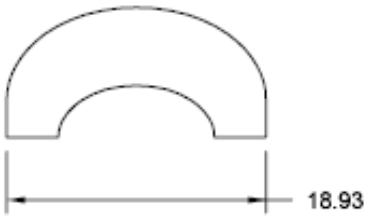
dimension text moved
back to home position

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.



text centered vertically on
the dimension line



result of moving text to the right
and outside the extension lines

Use Dimension Text Grips

Hover over a dimension text grip to quickly access the following functionality:

- **Stretch.** This is the default grip behavior:
  - If the text is positioned on the dimension line, *Stretch* moves the dimension line farther away or closer to the object being dimensioned. Use command line prompts to specify a different base point or copy the dimension line.

- If the text is positioned away from the dimension line, with or without a leader, *Stretch* moves the text without moving the dimension line.

- **Move with Dim Line.** Positions text on the dimension line, and moves the dimension line farther away or closer to the object being dimensioned (no additional prompts).
- **Move Text Only.** Positions the dimension text without moving the dimension line.
- **Move with Leader.** Positions the dimension text with a leader line to the dimension line.
- **Above Dim Line.** Positions the dimension text above the dimension line (left of the dimension line for vertical dimensions).
- **Center Vertically.** Positions the dimension text so that the dimension line cuts through the vertical center of the text.
- **Reset Text Position.** Positions the dimension text back to its default (or "home") position based on the active dimension style.

See also:

- Control Dimension Text on page 1543
- Overview of Modifying Dimensions on page 1593

To rotate dimension text

1. Annotate tab ➤ Dimensions panel ➤ Text Angle. 

2. Select the dimension to edit.
3. Enter the new angle for the text.

To return dimension text to its home position

1. Select the dimension text you want to return to its home position.
2. Hover over the text grip and click Rest Text Position on the grip menu.

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.

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.

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.

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.

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.

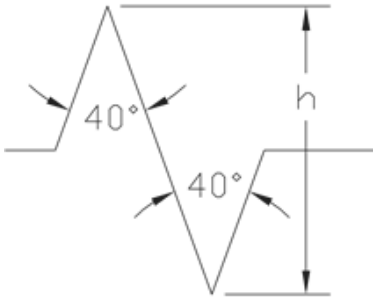
MTEXTED

Sets the application for editing multiline text objects.

Dimension Jog

Jog lines are used to represent a dimension value that does not display the actual measurement in a linear dimension. 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.

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.

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.

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.

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.

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

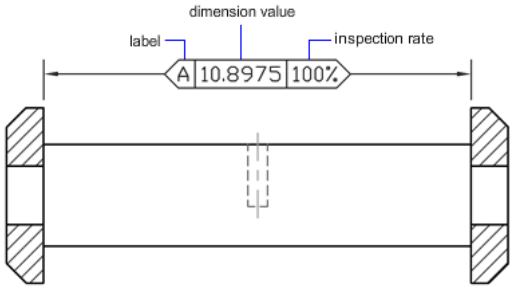
Creates a series of dimensions quickly from selected objects.

Convert Dimensions into Inspection Dimensions

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.

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.

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.

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.

Quick Reference

Commands

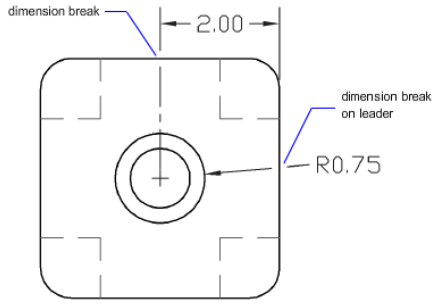
DIMINSPECT

Adds or removes inspection information for a selected dimension.

Break a Dimension Line

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, including aligned and rotated
- Angular dimensions, including 2- and 3-point
- Radial dimensions, including radius, diameter, and jogged
- Arc length dimensions
- Ordinate dimensions
- Multileaders that use straight-line leaders

The following dimension and leader objects do not support dimension breaks:

- Multileaders that use spline leaders
- Leaders created with the LEADER command

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 arrowhead 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 also remove dimension breaks from dimensions or multileaders with the Remove option of DIMBREAK. When removing them, *all* dimension breaks are removed from the selected dimension or multileader, but you can always add them back individually.

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 dimension breaks automatically, 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.

Dimension Break Gap Size

You control the size of dimension breaks on the New/Modify Dimension Style dialog box, Symbols and Arrows tab. 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 1324.

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.

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.

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.

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.

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.

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.

Quick Reference

Commands

DIMBREAK

Breaks or restores dimension and extension lines where they cross other objects.

DIMSTYLE

Creates and modifies dimension styles.

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.

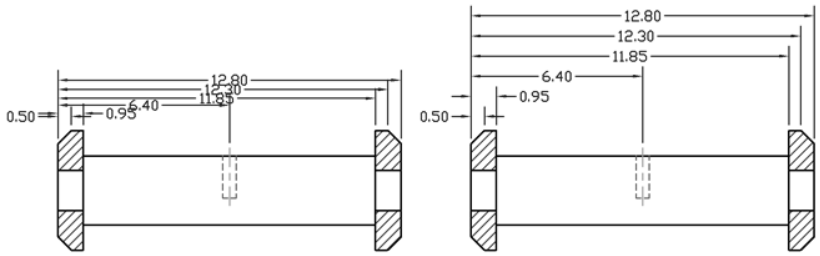
Parallel linear and angular dimensions can be created in a number of different ways in a drawing. With the DIMLINEAR and DIMANGULAR commands you can place one dimension at a time; you can use the DIMBASELINE and DIMCONTINUE commands to help place additional linear dimensions based on the previous linear 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.

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.

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.

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

Creates a series of dimensions quickly from selected objects.

System Variables

DIMDLI

Controls the spacing of the dimension lines in baseline dimensions.

Apply a New Dimension Style to Existing Dimensions

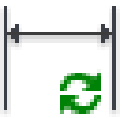
You can modify existing dimensions by applying a different dimension style. If you make changes to a dimension style, you can choose whether to update the dimensions associated with that dimension style.

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.

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.

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.

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 Override Current Style dialog box, 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.

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.

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.

Add Geometric Tolerances

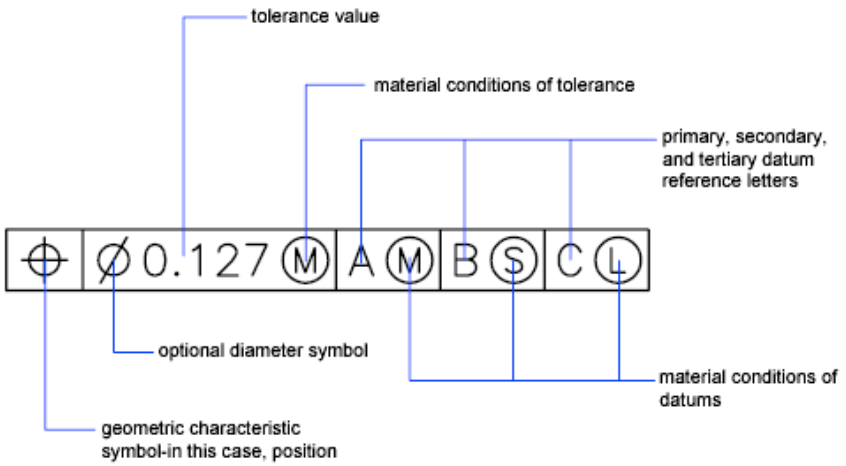
You can add geometric tolerances that show acceptable deviations of form, profile, orientation, location, and runout of a feature.

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 annotative on page 2289 tolerances. For more information about creating and working with an annotative tolerances, see Create Annotative Dimensions and Tolerances on page 1339.

See also:

- Scale Annotations on page 1324
- Create Annotative Dimensions and Tolerances on page 1339

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.

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.

Quick Reference

Commands

LEADER

Creates a line that connects annotation to a feature.

TOLERANCE

Creates geometric tolerances contained in a feature control frame.

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

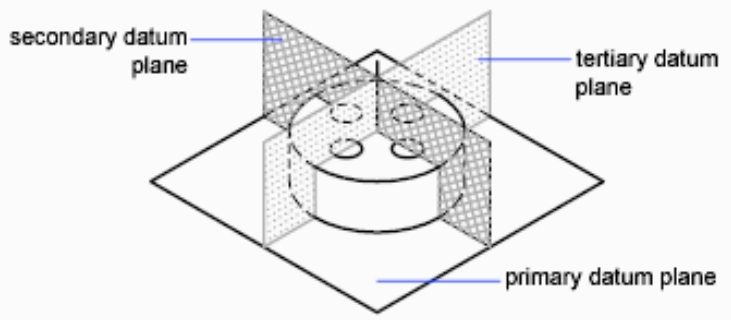
Creates geometric tolerances contained in a feature control frame.

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

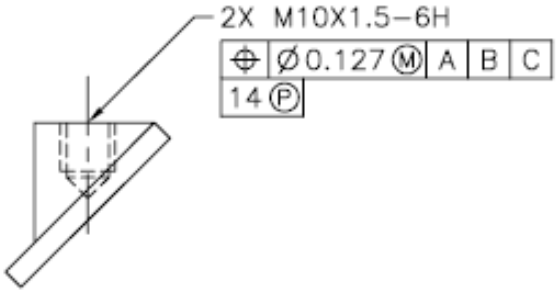
Creates geometric tolerances contained in a feature control frame.

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.

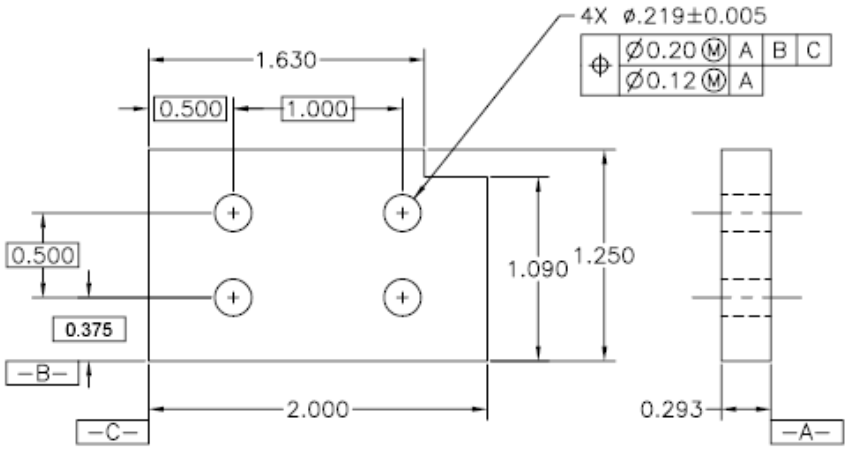
Composite Tolerances

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.

Part 10: Plot and Publish Drawings

Specify Settings for Plotting and Publishing

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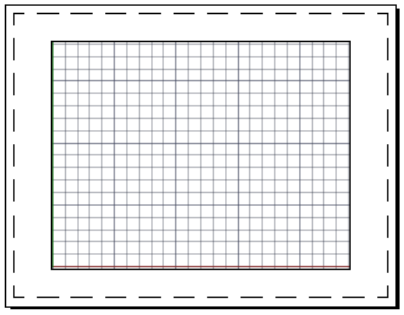
Before you plot or publish a drawing, you must specify the settings that determine the output. To save time, you can store these settings with the drawing as a *named page setup*.

Save Plot Settings as Named Page Setups

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.

Before you plot a drawing, you must specify the settings that determine the appearance and format of the output. To save time, you can store these settings with the drawing as a *named page setup*.

For example, when you access a layout for the first time, a single layout viewport is displayed, and a dashed line indicates the printable area of the paper for the currently configured paper size and printer or plotter.



In addition, the page setup also includes many other settings and options such as

- The orientation of the plot, portrait or landscape
- The plot scale
- Whether lineweights should be plotted
- The shading style

By default, the first time you access a layout, it becomes *initialized*, and a default page setup is assigned to it. Default page setups are assigned names such as *model*, *layout1*, *layout2*, and so on.

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.

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.

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.

Quick Reference

Commands

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.

PLOTTERMANAGER

Displays the Plotter Manager, where you can add or edit a plotter configuration.

PSETUPIN

Imports a user-defined page setup into a new drawing layout.

Reuse 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.

Named page setups are saved in the current drawing file and can be imported into other drawing files and applied to other layouts.

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 a named page setup to model space or to a layout using the Page Setup Manager. Other options available in the Page Setup Manager include

- Apply a named page setup saved with one layout to another layout in the same drawing
- Modify the settings of a page setup at any time
- Import a named page setup from another drawing, and apply it to layouts in the current drawing

You can also 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 the layout, it is plotted with the settings you specified.

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.

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.

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.

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.

3 Click Close.

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.

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.

PSETUPIN

Imports a user-defined page setup into a new drawing layout.

Specify Page Setup Settings

Page setups are associated with model space and with layouts, and are saved in the drawing file. The settings specified in a page setup determine the appearance and format of your final output.

Select a Printer or Plotter for a Layout

To print a layout, select a printing or plotting device in the Page Setup dialog box. You can also view details about the name and location of the device, and 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 1666
- "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.

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.

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.

PSETUPIN

Imports a user-defined page setup into a new drawing layout.

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.

NOTE 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.

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.

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.

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 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.

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.

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.

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.

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.

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.

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.

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 model space or a layout, 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 1667

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.

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.

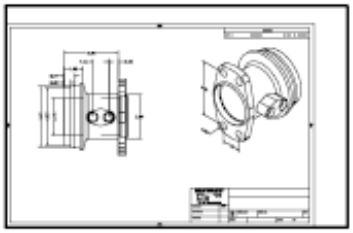
Adjust the Plot Offset of a Layout

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.

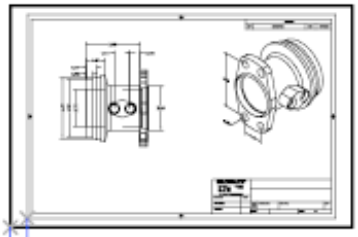
The plot offset specifies an offset of the plot area relative to the lower-left corner (the origin) of the printable area or the edge of the paper.

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.

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

PLOTOFFSET

Controls whether the plot offset is relative to the printable area or to the edge of the paper.

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 450
- 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.

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.

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.

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 574

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.

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.

Select a Plot Style Table for a Layout

A plot style table is a collection of plot styles assigned to a layout or model space.

A plot style is an object property, similar to linetype and color. Therefore, it can be assigned to an object or a layer and they control an object's plotted properties.

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 1674

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.

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.

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.

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.

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.

STYLESMANAGER

Displays the Plot Style Manager, where you can revise plot style tables.

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.

See also:

- Set Shaded Viewport Options on page 1677
- Set Options for Plotted Objects on page 1681

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.

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.

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.

Other Ways to Create Page Setups

There are several other ways to create named page setups.

Use the Layout Wizard to Specify Page Setups

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.

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.

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 or current named (paper space) layout.

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:

- Reuse Named Page Setups on page 1632

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.

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.

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.

Print or Plot Drawings 33

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.

Overview of Plotting

Understanding terms and concepts that relate to plotting makes your first plotting experience in the program easier.

Am I Printing or Plotting?

The terms *printing* and *plotting* can be used interchangeably for CAD output. Historically, printers would generate text only, and plotters would generate vector graphics. As printers became more powerful and could generate high-quality raster images of vector data, the distinction mostly disappeared.

In addition to paper output, electronic delivery of multiple drawing sheets uses the encompassing term, *publishing*. Also, the capability to generate physical models in plastic and metal is called *3D printing*.

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 that you want to set up.

Layouts

A layout represents a drawing sheet, and typically includes

- A drawing border and title block
- One or more layout viewports that display views of model space
- General notes, labels, and possibly dimensions
- Tables and schedules

Usually a drawing file contains only one layout, but you can create as many layouts as you need. The first time you display a layout, it is initialized and a default *page setup* is assigned to it.

Once initialized, layouts can be modified, published, and added to sheet sets as sheets.

Page Setups

When you create a layout, you specify a plotter, and settings such as paper size and orientation. These settings are saved in the drawing as a page setup. Each layout can be associated with a different page setup.

You can control these settings for layouts and for model space using the Page Setup Manager. You can name and save page setups for use with other layouts.

If you do not 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* is an optional method that controls how each object or layer is plotted. Assigning a plot style to an object or a layer *overrides* properties such as color, lineweight, and linetype when plotting. Only the appearance of plotted objects is affected by plot style.

Plot style tables collect groups of plot styles, and save them in a file that you can later apply when plotting.

The *Plot Style Manager* is a folder that contains all the available plot style tables, along with the Add-A-Plot-Style wizard.

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.

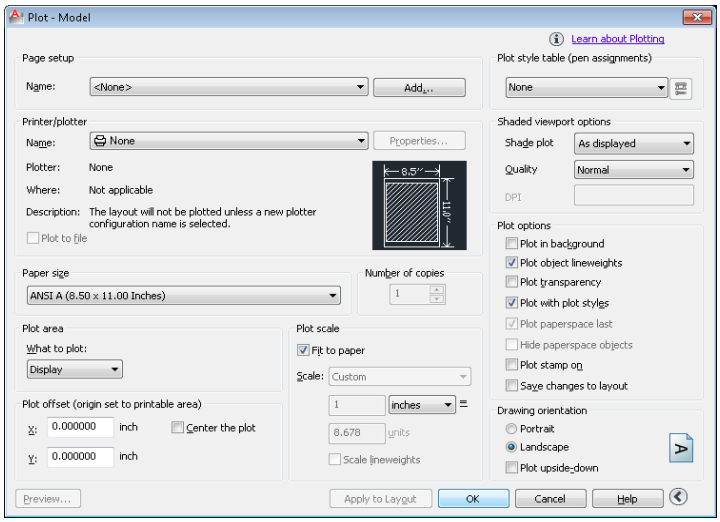
IMPORTANT A drawing file or drawing template file that was created with an educational version will always be plotted with the following plot stamp: PRODUCED BY AN AUTODESK EDUCATIONAL PRODUCT. Blocks and xrefs created with an educational version and used in a commercial version will also result in the educational plot stamp being plotted.

See also:

- Create Multiple-View Drawing Layouts (Paper Space) on page 431
- Specify Settings for Plotting and Publishing on page 1629
- "Use Plotters and Printers" 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.

10 Under Drawing Orientation, select an orientation.

11 Click OK.

To plot at the command prompt instead of in a dialog box, use -PLOT.

To turn background plotting on or off

1. Click Application 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.

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.

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 or current named (paper space) 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.

Use a Page Setup to Specify Plot Settings

You can use a page setup to save and reuse settings for your plot jobs.

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.

See also:

- Specify Settings for Plotting
- "Use Plotters and Printers" in the *Driver and Peripheral Guide*

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.

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.

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.

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.

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.

See also:

- Specify Settings for Plotting and Publishing on page 1629
- "Use Plotters and Printers" in the *Driver and Peripheral Guide*

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.

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.

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.

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.

Set Paper Size

When plotting a drawing, 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 space, 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.

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.

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.

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.

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

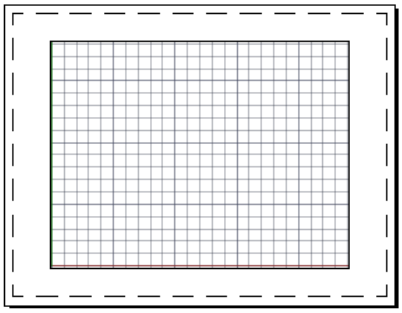
Controls the display of a warning dialog box when attempting to print a layout with a paper size different from the paper size specified by the default for the plotter configuration file.

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 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.

Set the Position of the Plot

The printable area of a drawing sheet is defined by the selected printer or plotter, but you can change the position of plot relative to the printable area or to the edge of the paper.

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 shift 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.

NOTE If you specify a different printer or plotter, the printable area might change.

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.

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.

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 model space or a layout:

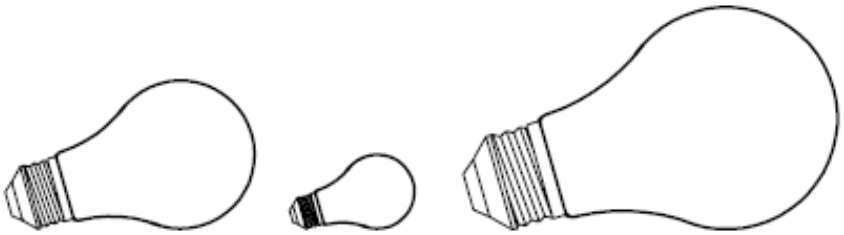
- From model space, you can establish the scale in the Plot dialog box. This scale represents a ratio of plotted units to the world-size units you used to draw the model.
- In a layout, you work with two scales. The first affects the overall layout of the drawing, which usually is scaled 1:1, based on 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.

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.

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.

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.

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.

Overview of Shaded Viewport Plotting

The options for shaded viewport plotting provide flexibility in presenting your 3D designs to others.

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.

NOTE If hardware acceleration is turned off, or it is turned on 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 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.

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.

GRAPHICSCONFIG

Sets options for 3D display performance.

PLOT

Plots a drawing to a plotter, printer, or file.

SHADEMODE

Starts the VSCURRENT command.

VSCURRENT

Sets the visual style in the current viewport.

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.

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.

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.

Quick Reference

Commands

3DCONFIG

Sets options that affect 3D display performance.

GRAPHICSCONFIG

Sets options for 3D display performance.

PLOT

Plots a drawing to a plotter, printer, or file.

SHADEMODE

Starts the VSCURRENT command.

VSCURRENT

Sets the visual style in the current viewport.

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.

Quick Reference

Commands

PLOT

Plots a drawing to a plotter, printer, or file.

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 Transparency.** Specifies that transparency levels applied to objects and layers are plotted. Plot Transparency applies to wireframe and hidden plots only. Other visual styles, such as Realistic, Conceptual, or Shaded will always plot with transparency.

  IMPORTANT This setting can be overridden by the PLOTTRANSPARENCYOVERRIDE system variable.

- **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 layout viewport. The effect of this setting is reflected in the plot preview, but not in the layout.
- **Plot Stamp On.** Turns on plot stamps and places a plot stamp on a specified corner of each drawing and can add it to a log 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.
- **Save Changes to Layout.** Saves changes you make in the Plot dialog box to the layout.

To set shaded plotting options from the Model tab

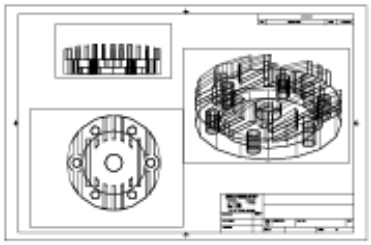
1. Click Output tab ➤ Plot panel ➤ Plot. 
2. In the Plot dialog box, Shaded Viewport Options area, Shade Plot drop-down list, select an option.

To set shaded plotting options from a layout tab

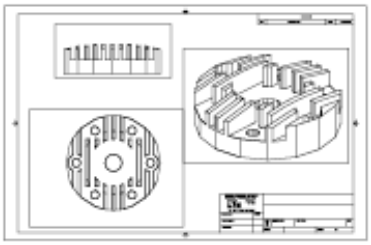
1. Select a layout viewport.
2. Double-click the viewport border to display the Properties palette.
3. On the Properties palette, click Shade Plot.
4. In the Shade Plot drop-down list, select an option.

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.



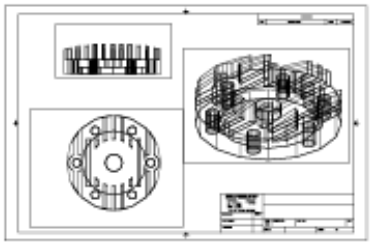
Hidden lines not removed



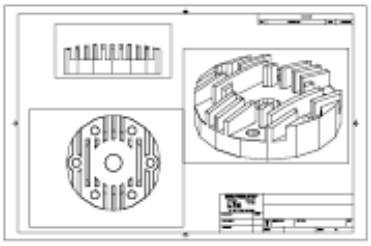
Hidden lines removed

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

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.

To plot transparency

Plot Transparency applies to wireframe and hidden plots only. Other visual styles, such as Realistic, Conceptual, or Shaded will always plot with transparency.

1. Click Output tab ➤ Plot panel ➤ Plot. 
2. In the Plot dialog box, under Plot Options, select Plot Transparency.
3. Click OK to plot the drawing.

WARNING Enabling this option may reduce plot performance, even if your drawing does not contain transparent objects. The PLOTTRANSPARENCYOVERRIDE system variable honors the Plot Transparency setting by default, but can be set to always or never plot transparency.

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.

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.

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.

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.

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 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
- Transparency
- 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.

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.

Quick Reference

Commands

OPTIONS

Customizes the program settings.

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.

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.

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.

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.

Quick Reference

Commands

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

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 Format ➤ Plot Style. 

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.

To change a plot style table description

1 Click Format ➤ Plot Style. 

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.

To edit plot styles in a plot style table

1 Click Format ➤ Plot Style. 

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.

Quick Reference

Commands

STYLESMANAGER

Displays the Plot Style Manager, where you can revise plot style tables.

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

Table	Description
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 1690

Quick Reference

Commands

STYLESMANAGER

Displays the Plot Style Manager, where you can revise plot style tables.

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.

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.

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.

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.

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 1702

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.

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.

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.

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.

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.

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.

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 1690

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.

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 Format ➤ Plot Style. 
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.

Quick Reference

Commands

STYLESMANAGER

Displays the Plot Style Manager, where you can revise plot style tables.

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 Format ➤ Plot Style. 
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.

Quick Reference

Commands

STYLESMANAGER

Displays the Plot Style Manager, where you can revise plot style tables.

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 Format ➤ Plot Style. 
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.

To use screening

1. Click Format ➤ Plot Style. 
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.

To enable or disable dithering

1. Click Format ➤ Plot Style. 
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.

To enable or disable conversion to grayscale

1. Click Format ➤ Plot Style. 
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.

Quick Reference

Commands

STYLESMANAGER

Displays the Plot Style Manager, where you can revise plot style tables.

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 Format ➤ Plot Style. 
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.

Quick Reference

Commands

STYLESMANAGER

Displays the Plot Style Manager, where you can revise plot style tables.

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 563
- Control Lineweights on page 574

To set the plotted lineweight

1. Click Format ➤ Plot Style. 
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.

To set the plotted linetype

1. Click Format ➤ Plot Style. 
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.

To apply scaling to non-ISO linetypes and fill patterns

1. Click Format ➤ Plot Style. 
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.

Quick Reference

Commands

STYLESMANAGER

Displays the Plot Style Manager, where you can revise plot style tables.

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 Format ➤ Plot Style. 
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.

Quick Reference

Commands

STYLESMANAGER

Displays the Plot Style Manager, where you can revise plot style tables.

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:

- Modify Hatches and Fills on page 1389

To assign a fill style

1. Click Format ➤ Plot Style. 
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.

Quick Reference

Commands

STYLESMANAGER

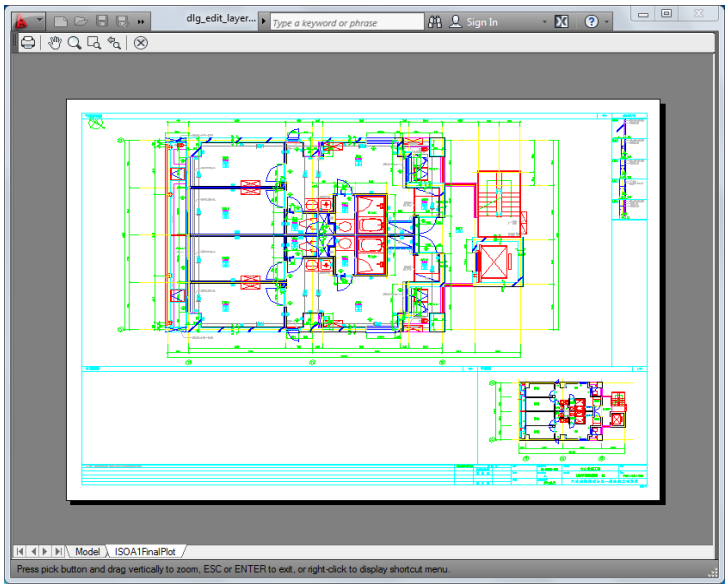
Displays the Plot Style Manager, where you can revise plot style tables.

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 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.

Quick Reference

Commands

PAN

Moves the view planar to the screen.

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.

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.

See also:

- Publish Drawings on page 1727

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 1727
- Review and Markup Files with Design Review on page 2119

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.

Quick Reference

Commands

PLOT

Plots a drawing to a plotter, printer, or file.

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 1727
- Review and Markup Files with Design Review on page 2119

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.

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.

Quick Reference

Commands

PLOT

Plots a drawing to a plotter, printer, or file.

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.

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.

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.

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.

Quick Reference

Commands

PLOT

Plots a drawing to a plotter, printer, or file.

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 on page 1951
- "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.

Quick Reference

Commands

PLOT

Plots a drawing to a plotter, printer, or file.

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.

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.

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.

Publish Drawings

34

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 1758.

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 495
- Specify Page Setup Settings on page 1635
- Publish 3D DWF Files on page 1758
- Review and Markup Files with Design Review on page 2119

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.

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.

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.

Quick Reference

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.

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.

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.

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.

NOTE 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.)

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 in the shortcut menu, 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 in the shortcut menu.

- **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 in the shortcut menu.

- **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.

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.

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.

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.

Quick Reference

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.

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 1753

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.

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

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.

PUBLISHCOLLATE

Controls whether plotting a sheet set, multi-sheet plot file, or plot spool file can be interrupted by other plot jobs.

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 1753
- Review and Markup Files with Design Review on page 2119

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.

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.

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

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.

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.

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 1655
- Work with Sheets in a Sheet Set on page 471
- Set Publish Options on page 1753

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.

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.

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.

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.

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.

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.

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.

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.

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.

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.

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.

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.

Quick Reference

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.

PUBLISHCOLLATE

Controls whether plotting a sheet set, multi-sheet plot file, or plot spool file can be interrupted by other plot jobs.

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.

Quick Reference

PUBLISH

Publishes drawings to DWF, DWFx, and PDF files, or to plotters.

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 on page 2119.

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 on page 2119

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

PUBLISH

Publishes drawings to DWF, DWFx, and PDF files, or to plotters.

Set Publish Options

Set options for publishing, such as output file location, multi-sheet name options, 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.

NOTE PDF files cannot be password protected.

To specify output locations and types, file names, and security settings for DWF, DWFx, PDF, and plot files

1. Click Output tab ➤ Plot panel ➤ Batch Plot.
2. In the Publish To group, select DWF Format; and select DWF File or DWFx File.
3. Click Publish Options.

To be prompted for a name for a multi-sheet DWF, DWFx, or PDF file

1. Click Output tab ➤ Plot panel ➤ Batch Plot.
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.

To include layer or block information in a published DWF, DWFx, or PDF file

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 ➤ Plot panel ➤ Batch Plot.

2 Click Publish Options.

The Publish Options dialog box appears.

3 In the DWF Data Options, do one of the following:

- Under Layer Information, click to display the drop-down list, and select Include.
- Under Block Information, click to display the drop-down list, and select Include.

NOTE By default, Layer and Block Information is set to Don't Include. If you change the setting to include layer or block information, you can turn individual layers on and off when you view or print the DWF file, or 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.

To create a block template (BLK) file

1 Click Output tab ➤ Plot panel ➤ Batch Plot.

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.

To edit a block template (BLK) file

1 Click Output tab ➤ Plot panel ➤ Batch Plot.

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.

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.

Quick Reference

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.

AUTODWFPUBLISH

Controls whether DWF (Design Web Format) files are created automatically when you save or close drawing (DWG) files.

AUTOMATICPUB

Controls whether electronic files (DWF/PDF) 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.

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	

Unsupported Content	Details
Block attributes	
Font types (Various)	See the "Supported Text Fonts" table
Gradient fills (Hatches)	
Hidden edges	
Hyperlinks	
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
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 on page 2119

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.

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.

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.

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.

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.

7 In 3D DWF options group, click Yes to publish materials. Click OK.

8 Click 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.

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.

7 In 3D DWF options group, click Yes to publish materials. Click OK.

8 Click 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.

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

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.

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.

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 print 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 optimizing models.

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

Quick Reference

3DPRINT

Sends a 3D model to a 3D printing service.

SMOOTHMESHCONVERT

Sets whether mesh objects that you convert to 3D solids or surfaces are smoothed or faceted, and whether their faces are merged.

Part 11: Share Data Between Files

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Reference Other Drawing Files

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Attached xrefs are linked to, but not actually inserted in, another drawing. Therefore, with xrefs you can build drawings without significantly increasing the drawing file size.

See also:

■

Overview of Referenced Drawings (Xrefs)

By using referenced drawings, you can

- Coordinate your work with the work of others by referencing other drawings in your drawing to keep up with the changes being made by other designers. You can also assemble a master drawing from component drawings that may undergo changes as a project develops.
- Ensure that the most recent version of the referenced drawing is displayed. When you open your drawing, each referenced drawing is automatically reloaded, so it reflects the latest state of the referenced drawing file.
- Keep the names of layers, dimensioning styles, text styles, and other named elements in your drawing separate from those in referenced drawings.
- Merge (bind) attached referenced drawings permanently with your current drawing when the project is complete and ready to be archived.

NOTE Like a block reference, an xref appears in the current drawing as a single object. However, you cannot explode an xref without binding it first.

See also:

- DesignCenter on page 84

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.

EXTERNALREFERENCES

Opens the External References palette.

EXTERNALREFERENCESCLOSE

Closes the External References palette.

REFCLOSE

Saves back or discards changes made during in-place editing of a reference, either an xref or a block definition.

REFEDIT

Edits an xref or a block definition directly within the current drawing.

REFSET

Adds or removes objects from a working set during in-place editing of a reference, either an xref or a block definition.

RENAME

Changes the names assigned to items such as layers and dimension styles.

XATTACH

Inserts DWG files as an external reference (xref).

XBIND

Binds one or more definitions of named objects in an xref to the current drawing.

XCLIP

Crops the display of a selected external reference or block reference to a specified boundary.

XREF

Starts the EXTERNALREFERENCES command.

System Variables

BINDTYPE

Controls how xref names are handled when binding xrefs or editing xrefs in place.

ERSTATE

Indicates whether the External References palette is open or closed.

INDEXCTL

Controls whether layer and spatial indexes are created and saved in drawing files.

INSBASE

Stores the insertion base point set by BASE, which gets expressed as a UCS coordinate for the current space.

PROJECTNAME

Assigns a project name to the current drawing.

VISRETAIN

Controls the properties of xref-dependent layers.

XCLIPFRAME

Determines whether xref clipping boundaries are visible or plotted in the current drawing.

XEDIT

Controls whether the current drawing can be edited in-place when being referenced by another drawing.

XLOADCTL

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

XLOADPATH

Creates a path for storing temporary copies of demand-loaded xref files.

XREFNOTIFY

Controls the notification for updated or missing xrefs.

Attach and Detach Referenced Drawings

You can perform several operations on referenced drawing files (xrefs).

Attach Drawing References (Xrefs)

You can insert any drawing file as an external reference or *xref* in the current drawing.

When you attach a drawing file as an xref, you link that referenced drawing to the current drawing. Any changes to the referenced drawing are displayed in the current drawing when it is opened or reloaded.

A drawing file can be attached as an xref to multiple drawings at the same time. Conversely, multiple drawings can be attached as referenced drawings to a single drawing.

Tools for Attaching Xrefs

You can use several methods to attach an xref:

- Click View tab ➤ Palettes panel ➤ External References Palette.
- At the Command prompt, enter **externalreferences**
- At the Command prompt, enter **xattach**.

You can also use DesignCenter™ to attach xrefs to a drawing. Use DesignCenter for simple attachments, previewing drawing references and their descriptions, and quick placement by dragging.

You can attach an xref by dragging it from DesignCenter or by clicking Attach as Xref on the shortcut menu.

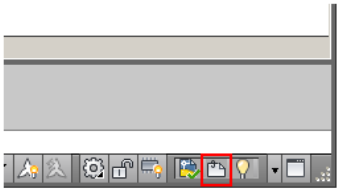
The saved path used to locate the xref can be a full path, a relative (partially specified) path, or no path.

If an xref contains any variable block attributes, they are ignored.

NOTE When using the External References palette, it is recommended that you turn on the Auto-hide feature or anchor the palette. The palette will then hide automatically when you specify the insertion point of the external reference.

Receive Notification of Attached Xrefs

An xref icon is displayed in the lower-right corner of the application window (the status bar tray) when xrefs are attached to the drawing.

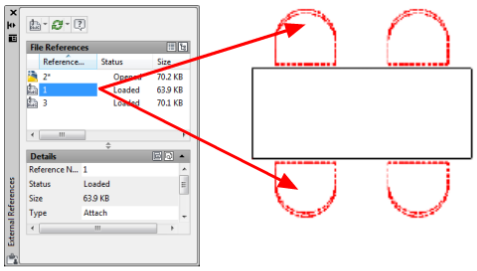


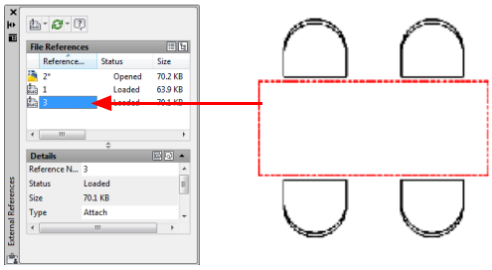
When one or more xrefs are not found or if any of the xrefs need reloading, an exclamation point is added to the Xref icon. If you click the Xref icon, the External References palette is displayed.

Highlight External References in a Drawing

To find an external reference in a complex drawing, select an item in the External References palette to highlight all visible instances in the drawing. Conversely, select an external reference in the drawing to highlight its name in the External References palette.

NOTE The ERHIGHLIGHT system variable controls whether referenced objects are highlighted. You can turn highlighting off to improve performance.





Control the Properties of Referenced Layers

You can control the visibility, color, linetype, and other properties of an xref's layers and make these changes temporary or permanent. If the VISRETAIN system variable is set to 0, these changes apply only to the current drawing session. They are discarded when you end the drawing session, or when you reload or detach the xref.

You can also control the fade display of the DWG xref. The XDWGFADECTL system variable defines the fade percentage for all DWG xrefs.

Xref Clipping Boundaries

Drawings can include xrefs that are clipped. If you want to see the clipping boundary, you can turn on the XCLIPFRAME system variable.

Attachments from Educational Products

If you open, insert, or attach an xref from an Autodesk Educational Product, the drawings you plot contain the following banner: "PRODUCED BY AN AUTODESK EDUCATIONAL PRODUCT."

See also:

- Nest and Overlay Referenced Drawings on page 1780
- Clip External References and Blocks on page 1793
- Set Paths to Referenced Drawings on page 1782
- DesignCenter on page 84
- Specify the Behavior of Dockable Windows on page 175

To attach an xref

1. Click Insert tab ➤ Reference panel ➤ Attach. 
2. In the Select Reference File dialog box, select one or more files you want to attach and then click Open.
3. In the Attach External Reference dialog box, under Reference Type, select Attachment.
4. Specify the insertion point, scale, and rotation angle. Click Specify On-Screen to use the pointing device.

   Attachment includes all nested xrefs.
5. Click OK.

To attach or overlay an xref using DesignCenter

1. Click View tab ➤ Palettes panel ➤ DesignCenter. 
2. In the content area or in the Search dialog box, locate the DWG reference you want to attach or overlay.
3. Right-click. Drag the DWG reference into the open drawing.
4. Release the right-pointing device button. Click Attach as Xref.
5. In the External Reference dialog box, under Reference Type, select Attachment or Overlay.
6. Enter values for Insertion Point, Scale, and Rotation, or select Specify On-Screen to use the pointing device.

7 Click OK.

You can also attach an xref by dragging or by clicking Attach as Xref on the shortcut menu.

8 Click OK.

To view referenced files by type in the External References palette

1 Click Insert tab ➤ Reference panel ➤ Dialog box launcher. 

2 In the External References palette, in the File References pane, click the List View button.

3 Click the Icon column.

The attached referenced files are sorted by type in the list view.

To adjust the fade display of the DWG xref

1 Click Insert tab ➤ Reference panel ➤ Xref Fading. 

2 Drag the slider to adjust the fade display of the DWG xref.

The DWG xref fades as you drag the slider.

Quick Reference

Commands

ADCENTER

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

EXTERNALREFERENCES

Opens the External References palette.

EXTERNALREFERENCESCLOSE

Closes the External References palette.

TRAYSETTINGS

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

XATTACH

Inserts DWG files as an external reference (xref).

XREF

Starts the EXTERNALREFERENCES command.

System Variables

ERHIGHLIGHT

Controls whether reference names or reference objects are highlighted when their counterparts are selected in the External References palette or in the drawing window.

ERSTATE

Indicates whether the External References palette is open or closed.

INDEXCTL

Controls whether layer and spatial indexes are created and saved in drawing files.

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.

VISRETAIN

Controls the properties of xref-dependent layers.

XDWGFADECTL

Controls the dimming for all DWG xref objects.

XLOADCTL

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

XLOADPATH

Creates a path for storing temporary copies of demand-loaded xref files.

XREFNOTIFY

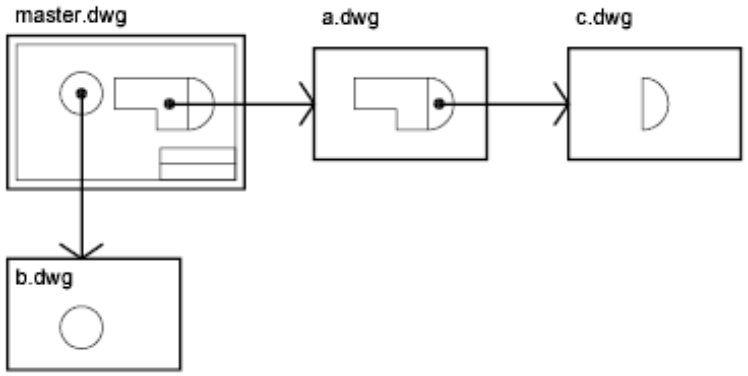
Controls the notification for updated or missing xrefs.

Nest and Overlay Referenced Drawings

Attached DWG references (xrefs) can be nested: that is, you can attach an xref that contains another xref.

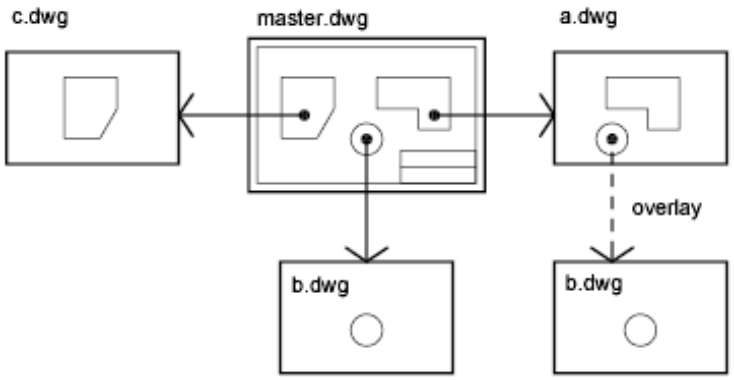
Xrefs can be nested within other xrefs: that is, you can attach an xref that contains another xref. You can attach as many copies of an xref as you want, and each copy can have a different position, scale, and rotation.

In the following illustration, *master.dwg* references *a.dwg* and *b.dwg*. Drawing *a.dwg* references *c.dwg*. In *master.dwg*, *c.dwg* is a nested xref.



You can also overlay an xref on your drawing. Unlike an attached xref, an overlaid xref is *not* included when the drawing is itself attached or overlaid as an xref to another drawing. Overlaid xrefs are designed for data sharing in a network environment. By overlaying an xref, you can see how your drawing relates to the drawings of other groups without changing your drawing by attaching an xref.

In the following illustration, several people are working on drawings referenced by *master.dwg*. The person working on *a.dwg* needs to see the work being completed by the person working on *b.dwg*, but does not want to xref *b.dwg* because it would then appear twice in *master.dwg*. Instead, the person overlays *b.dwg*, which is not included when *a.dwg* is referenced by *master.dwg*.



NOTE When using the parametric drawing feature, you can only constrain objects in the drawing to the insertion point of an Xref, and not its nested objects.

Relative Saved Paths and Nested Xrefs

The saved path for an xref can be a full path, a relative (partially specified) path, or no path. For a nested xref, a relative path always references the location of its immediate host and not necessarily the currently open drawing.

See also:

■

To overlay an xref

1. Click Insert tab ➤ Reference panel ➤ Attach. 
2. In the Select Reference File dialog box, select the file you want to attach and then click Open.
3. In the External Reference dialog box, under Reference Type, select Overlay.
4. Specify the insertion point, scale, and rotation angle. Click Specify On-Screen to use the pointing device.

   Using the Overlay option excludes any nested xrefs.
5. Click OK.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the External References palette.

System Variables

XREFTYPE

Controls the default reference type when attaching or overlaying an external reference.

Set Paths to Referenced Drawings

You can view and edit the file name and path used when locating a particular drawing reference (xref). Use this option if the referenced file has been moved to a different folder or renamed since it was first attached.

You can choose from three types of folder path information to save with an attached reference: a full path, a relative path, and no path.

Specify a Full (Absolute) Path

A full path is a fully specified hierarchy of folders that locates the file reference. A full path includes a local hard drive letter, a URL to a website, or a network server drive letter.

Specify a Relative Path

Relative paths are partially specified folder paths that assume the current drive letter or the folder of the host drawing. This is the most flexible option, and enables you to move a set of drawings from your current drive to a different drive that uses the same folder structure.

If the file that is being referenced is located on a different local hard drive or on a network server, the relative path option is not available.

The conventions for specifying a relative folder path are as follows:

\ Look in the root folder of the host drawing's drive

path From the folder of the host drawing, follow the specified path

path From the root folder, follow the specified path

.*path* From the folder of the host drawing, follow the specified path

..*path* From the folder of the host drawing, move up one folder level and follow the specified path

..\..*path* From the folder of the host drawing, move up two folder levels and follow the specified path

---

NOTE If a drawing that contains referenced files is moved or saved to a different path, to a different local hard drive, or to a different network server, you must edit any relative paths to accommodate the host drawing's new location or you must relocate the referenced files.

Specify No Path

When no path information is saved with the attached external reference, the following search is initiated in the order shown:

- Current folder of the host drawing
- Project search paths defined on the Files tab in the Options dialog box and in the PROJECTNAME system variable
- Support search paths defined on the Files tab in the Options dialog box
- Start In folder specified in the Microsoft® Windows® application shortcut

Specifying the No Path option is useful when moving a set of drawings to a different folder hierarchy or to an unknown folder hierarchy.

Know when a Referenced Drawing has been Relocated

If the drawing you are working on contains an xref that has been moved to a different folder, a message is displayed at the site of the xref when you load the drawing. The message indicates that the xref cannot be loaded using the old path. When you specify the new path, the xref is reloaded into your drawing.

Use Project Names to Define Referenced Drawing Paths

Project names make it easier for you to manage xrefs when drawings are exchanged between customers or if you have different drive mappings to the same location on a server. The project name points to a section in the registry that can contain one or more search paths for each project name defined.

If the program cannot find an xref at the location specified by the search path, the prefix (if any) is stripped from the path. If the drawing has a

PROJECTNAME value set and a corresponding entry exists in the registry, the program searches for the file along the project search paths. If the xref still is not located, the program search path is searched again.

You can add, remove, or modify the project names that exist in the registry. The folder search paths beneath the project name can also be added, removed, or modified.

The search paths beneath the project name can be added, removed, or modified in the same manner as the project name. The order in which the folders are searched can also be modified. Projects and their search paths can only be edited through the Files tab in the Options dialog box. You cannot edit project names at the Command prompt.

Once you have established a project name and the search paths you want associated with that project name, you can make that project name the currently active project. The program searches the paths associated with that currently active project for xrefs that were not found in the current folder of the host drawing, the full search path, the current drawing folder, or the program support paths.

Change Reference Paths in Multiple Drawings (Advanced)

When you relocate drawing files or the external files that they reference, including other drawing files, text fonts, images, and plot configurations, the reference paths that are saved in the drawing need to be updated. The Autodesk Reference Manager lists referenced files in selected drawings and provides tools to modify the saved reference paths without having to open each drawing file. You can also use Reference Manager to identify and fix unresolved references.

Reference Manager is a stand-alone application with its own Help system. To access it, click Start menu (Windows) ➤ (All) Programs ➤ Autodesk ➤ AutoCAD ➤ Reference Manager. If you are working on a network, it may not have been installed on your computer.

WARNING Reference Manager is intended primarily for CAD managers who thoroughly understand paths. It changes multiple paths at once, and you cannot undo your actions. Use extreme caution, especially if you are working on a network.

To change the path of a DWG reference

1. Click Insert menu ➤ External References.
2. In the External References palette, select a DWG reference name.

3 Under Found At, do one of the following:

- Edit the xref path directly.
- Click within the edit box, and then click the Browse button (...) that appears. Select the xref in its new path.

4 Click OK.

The program reloads the xref and then regenerates the drawing with the xref in place.

To display the currently defined project names

1 Click Tools menu ➤ Options.

2 In the Options dialog box, Files tab, double-click Project Files Search Path.

3 Click each project name folder to display the search paths associated with it.

4 Click OK (or Apply).

To add a project name

1 Click Tools menu ➤ Options.

2 In the Options dialog box, Files tab, double-click Project Files Search Path. Click Add.

A folder named *projectx* (where *x* indicates the next available number) is created and indented beneath the project folder.

3 Either enter a new name, or press Enter to accept *projectx*.

The project name must be 31 characters or fewer, and it cannot contain leading spaces or terminating spaces.

4 Click OK (or Apply).

To remove a project name

1 Click Tools menu ➤ Options.

2 In the Options dialog box, Files tab, double-click Project Files Search Path.

3 Select a project name. Click Remove.

4 Click OK (or Apply).

To modify a project name

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Files tab, select a project name. Enter a new name.
3. Click OK (or Apply).

   You can also modify a project name by selecting the name in the project folder and pressing F2.

To add a search path

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Files tab, select a project name. Click Add.
3. Add a new search path beneath the project name by entering a new path, or click Browse and select a new path.
4. Click OK (or Apply).

   The new path is indented and placed beneath the project name.

To delete a search path

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Files tab, select a project name. Click Remove.
3. Click OK.

To change a search path

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Files tab, select a project name. Click Browse.
3. In the Browse for Folder dialog box, select a new path.
4. Click OK to close each dialog box.

   You can also change a search path by selecting the project path and pressing F2.

To make a project current

1. Click Tools menu ➤ Options.

2 In the Options dialog box, Files tab, double-click Project Files Search Path.

3 Select a Project name. Click Set Current.

4 Click OK (or Apply).

You can also set a project current by entering **projectname** at the Command prompt and then entering the name of the project.

To clear the current project

1 Click Tools menu ➤ Options.

2 In the Options dialog box, Files tab, click Project Files Search Path.

3 Click Clear Current.

This clears the setting for the current drawing.

4 Click OK (or Apply).

You can also clear the current project at the Command prompt by entering **projectname** and then entering a period (.).

Quick Reference

Commands

XREF

Starts the EXTERNALREFERENCES command.

System Variables

PROJECTNAME

Assigns a project name to the current drawing.

Detach Referenced Drawings

To completely remove DWG references (xrefs) from your drawing, you need to detach them rather than erase them.

Erasing xrefs does not remove, for example, layer definitions associated with those xrefs. Using the Detach option removes the xrefs and all associated information.

To detach an xref

1. Click View tab ➤ Palettes panel ➤ External References . 
2. In the External Reference palette, select a DWG reference.
3. Right-click the selected DWG reference and select Detach from the shortcut menu.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the External References palette.

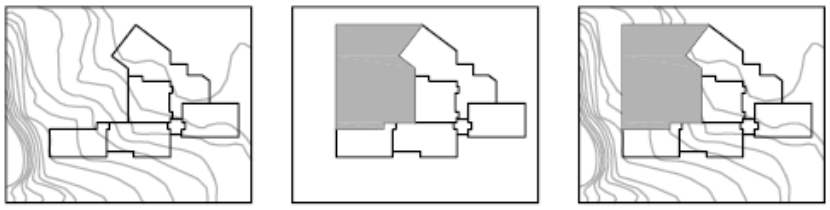
Update and Archive Referenced Drawings

You can update referenced drawings (xrefs) to make sure that they are current, and you can choose how xrefs are treated when a drawing is archived.

Update Referenced Drawing Attachments

When you open a drawing, all drawing references (xrefs) update automatically. You can also update xrefs whenever you want to ensure that the most current versions are displayed in your drawing.

When you open a drawing, all xrefs update automatically. Use the Reload option from the External References palette to update xrefs whenever you want to ensure that the most current versions are displayed in your drawing.



drawing with attached xref

referenced drawing is changed

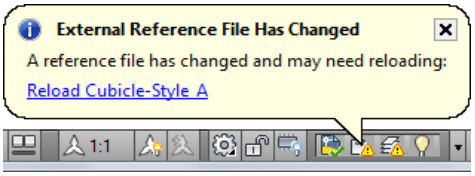
drawing with reloaded xref

Whenever you modify and save an externally referenced drawing in a network environment, other people can access your changes immediately by reloading the xrefs in their open drawings.

Receive Notification of Changed Xrefs

When you attach xrefs to a drawing, the program periodically checks whether the referenced files have changed since the last time the xrefs were loaded or reloaded. The XREFNOTIFY system variable controls xref notification.

By default, if a referenced file has changed, a balloon message is displayed near the Xref icon in the lower-right corner of the application window (the status bar tray). Click the link in the balloon to reload all changed xrefs.



If you close the balloon message without reloading, an exclamation point is added to the Xref icon. If you click the Xref icon, the External References palette is displayed.

By default, the program checks for changed xrefs every five minutes. You can change the number of minutes between checks by setting the XNOTIFYTIME system registry variable using **(setenv** "**XNOTIFYTIME**" "**n**"**)** where *n* is a number of minutes between 1 and 10080 (seven days).

NOTE When changing the value of XNOTIFYTIME, you must enter **XNOTIFYTIME** with the capitalization as shown.

Update Xrefs with Demand Loading Turned On

If demand loading is turned on when you load or reload an xref

- With the XLOADCTL system variable set to 1, the referenced drawing is kept open and locked. No one else can modify the referenced drawing.
- With XLOADCTL set to 2, a temporary copy of the most recently saved version of the referenced file is opened and locked. Others can open and modify the referenced drawing.

For information about demand loading, see Increase Performance with Large Referenced Drawings on page 1815.

To update an attached xref

1. Click Insert menu ➤ External References.
2. In the External References palette, select the reference name that you want to reload.
3. Right-click, and click Reload.

NOTE If the drawing you selected has been changed since you opened your drawing, the xref is reloaded.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the External References palette.

EXTERNALREFERENCESCLOSE

Closes the External References palette.

TRAYSETTINGS

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

System Variables

INDEXCTL

Controls whether layer and spatial indexes are created and saved in drawing files.

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.

XEDIT

Controls whether the current drawing can be edited in-place when being referenced by another drawing.

XLOADCTL

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

XLOADPATH

Creates a path for storing temporary copies of demand-loaded xref files.

XREFNOTIFY

Controls the notification for updated or missing xrefs.

Archive Drawings That Contain Referenced Drawings (Bind)

When you archive final drawings that contain xrefs, you can choose how you store the xrefs in the drawings.

When you archive final drawings that contain xrefs, you have two choices:

- Store the xref drawings along with the final drawing
- Bind the xref drawings to the final drawing

Storing an xref drawing along with the final drawing requires that the drawings always remain together. Any change to the referenced drawing will continue to be reflected in the final drawing.

To prevent unintentional updating of archived drawings by later changes to referenced drawings, bind the xrefs to the final drawing.

Binding an xref to a drawing makes the xref a permanent part of the drawing and no longer an externally referenced file. You can bind the entire database of the xref drawing, including all its xref-dependent named objects (blocks, dimension styles, layers, linetypes, and text styles), by using the XREF Bind option. For more information, see Resolve Name Conflicts in External References on page 1810.

Binding xrefs to a drawing is also an easy way to send a drawing to reviewers. Rather than sending a master drawing plus each of the drawings it references, you can use the Bind option to merge the xrefs into the master drawing.

NOTE You cannot bind xrefs that contain proxy objects. For more information, see Work with Custom and Proxy Objects on page 1966.

To bind an xref to the current drawing

1. Click Insert tab ➤ Reference panel ➤ Dialog box launcher. 
2. In the External References palette, select the reference name that you want to bind.
3. Right-click, and click Bind.
4. In the Bind Xrefs dialog box, select one of the following options:
   - Bind converts the objects in the xref into a block reference. Named object definitions are added to the current drawing with a prefix of *blockname* n, where *n* is a number starting at 0.
   - Insert also converts the objects in the xref into a block reference. Named object definitions are merged into the current drawing without adding prefixes.
5. Click OK to close each dialog box.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the External References palette.

XBIND

Binds one or more definitions of named objects in an xref to the current drawing.

System Variables

BINDTYPE

Controls how xref names are handled when binding xrefs or editing xrefs in place.

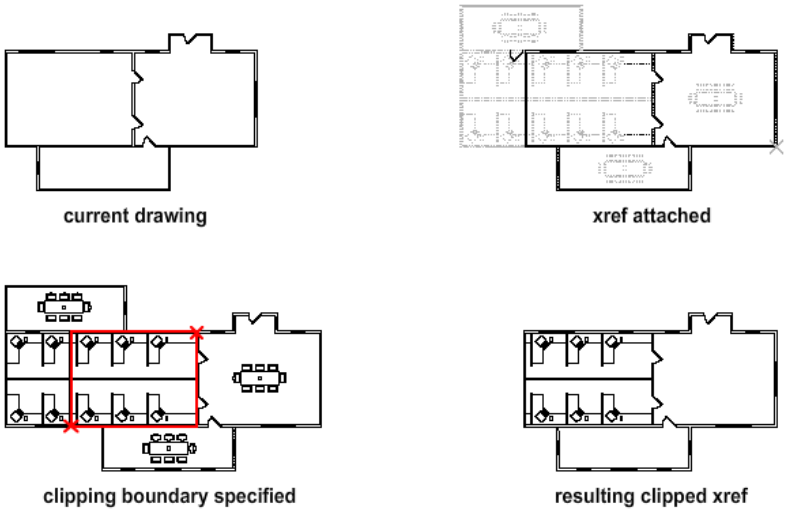
XEDIT

Controls whether the current drawing can be edited in-place when being referenced by another drawing.

Clip External References and Blocks

You can specify clipping boundaries to display a limited portion of an external reference drawing or block reference.

You can clip external references such as DGN, DWF, IMAGE, PDF underlays, or block references. With a clipping boundary, you can determine the portions of an external reference or block reference that you want to display by hiding the redundant parts of the reference inside or outside the boundary.



The clipping boundary can be a polyline, rectangle, or a polygon with vertices within the boundaries of the image. You can change the boundary of a clipped image. When you clip a boundary, the objects in the external reference or block are not altered; only their display is changed.

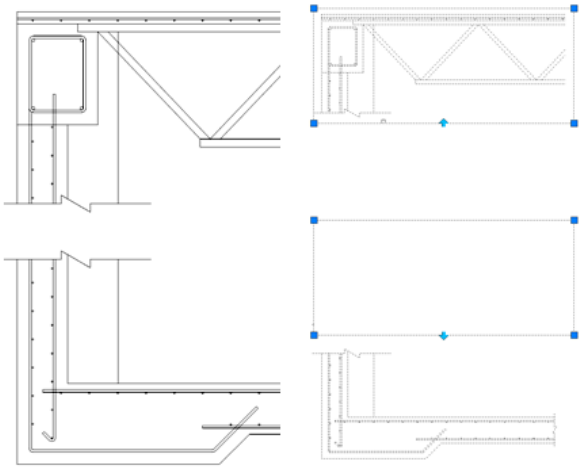
With the XCLIP, DGNCLIP, DWFCLIP, PDFCLIP, and IMAGECLIP commands, you can control the following viewing options:

Control the visibility of the clipped area of the external reference or block reference. When clipping is turned off, the boundary is not displayed and the entire external reference or block is visible, provided that the objects are on layers that are turned on and thawed.

Clipping results can be turned on or off using the clipping commands. This controls whether the clipped area is hidden or displayed.

Control the visibility of clipping boundaries. You can control the display of the clipping boundary with a clipping frame. The clipping system variable for XREF, PDF, DGN, DWG, and IMAGE underlays are XCLIPFRAME, PDFFRAME, DGNFRAME, DWGFRAME, and IMAGEFRAME respectively.

Invert the area to be hidden, inside or outside the clipping boundary When you want the hidden parts of the clipped reference displayed or vice versa, use the grips to alter the display of the external reference or blocks. With grips located at the midpoint on the first edge of the clipping boundary, you can invert the display of the clipped reference inside or outside the boundary.



The grips are visible and can be used when the clipping system variable is turned on, the reference is selected, and clipped.

Editing Options

After an external reference or block reference has been clipped, it can be moved, copied, or rotated just like an unclipped external reference or block reference. The clipping boundary moves with the reference. If an xref contains nested clipped xrefs, they appear clipped in the drawing. If the parent xref is clipped, the nested xrefs are also clipped.

Resize Clipping Boundaries

If you want to change the shape or size of a clipping boundary for external references and block references, you can use grips to edit the vertices just as you edit any object with grips.

In case of rectangular grip editing, you can maintain the closed four-sided rectangle or square shape of the rectangular clipping boundary because two vertices of the same side of the rectangular clipping boundary are edited together.

NOTE With Clipping boundaries, you cannot display self-intersecting polygonal boundaries. An error message is displayed and the boundary reverts to the last boundary.

Limitations for Clipping Boundaries

When clipping an referenced drawing or block the following limitations apply:

- A clipping boundary can be specified anywhere in 3D space, but it is always applied planar to the current UCS.
- If a polyline is selected, the clipping boundary is applied in the plane of that polyline.
- Images in external references or blocks are always clipped within the rectangular extents of the reference. When you apply polygonal clipping to images in externally referenced drawings, the clipping boundary is applied to the rectangular extents of the polygonal boundary, rather than to the polygon itself.

See also:

- Clip Raster Images on page 1924
- Clip Underlays on page 1903

To clip an xref

1. Click Insert tab ➤ Reference panel ➤ Clip Xref.
2. Select an xref.
3. At the prompt, specify New Boundary by pressing Enter.
4. Select a polyline or specify a rectangular or polygonal clipping boundary. Specify the corners or vertices of the boundary.

5 (Optional) Use the Invert Clip option to change the area to be hidden from outside to inside the clipping boundary.

The xref is clipped based on the area that you specified.

NOTE You can use DGNCLIP, DWFCLIP, PDFCLIP, or IMAGECLIP to clip their corresponding xref attachments: DGNATTACH, DWFATTACH, PDFATTACH, or IMAGEATTACH.

To invert a clipped reference

1. Select the external reference or block reference to be clipped.
2. Set XCLIPFRAME, DWFFRAME, DGNFRAME, IMAGEFRAME, or PDFFRAME to 1. This specifies that the clipping boundary is displayed.
3. Select the clipped boundary to see the grip at the midpoint on the first edge of the clipped boundary.
4. Click the grip to invert the display of the reference clipped inside or outside the boundary.

Quick Reference

Commands

DGNCLIP

Crops the display of a selected DGN underlay to a specified boundary.

DWFCLIP

Crops the display of a selected DWF or DWFx underlay to a specified boundary.

IMAGECLIP

Crops the display of a selected image to a specified boundary.

PDFCLIP

Crops the display of a selected PDF underlay to a specified boundary.

XCLIP

Crops the display of a selected external reference or block reference to a specified boundary.

System Variables

DGNFRAME

Determines whether DGN underlay frames are visible or plotted in the current drawing.

DWFFRAME

Determines whether DWF or DWFx underlay frames are visible or plotted in the current drawing.

FRAMESELECTION

Controls whether the frame of an image, underlay, or clipped xref can be selected.

IMAGEFRAME

Controls whether image frames are displayed and plotted.

XCLIPFRAME

Determines whether xref clipping boundaries are visible or plotted in the current drawing.

Edit Referenced Drawings

Referenced drawings can be edited by opening them directly, or you can edit the xref in place from within the current drawing. You can edit a block definition directly from any selected block reference.

Edit a Referenced Drawing in a Separate Window

While the simplest and most direct method for editing xrefs is to open the source file for the referenced drawing, there is an alternative that can be more convenient.

If you need to edit the *model space* objects in an xref, you can access the xref or a nested xref directly from the External References palette or with the XOPEN command. Select the xref, and then using the shortcut menu in the palette, open its source file in a separate window. After you save the edits, close the source drawing, save back the edited xrefs on page 1804 in your original drawing, and resume working.

NOTE Make sure you know whether the referenced drawing is also referenced by other drawings, and the the changes you make are appropriate in other instances.

See also:

- Edit Referenced Drawings and Blocks with Nesting, OLE, or Attributes on page 1806
- Edit Selected Objects in Referenced Drawings and Blocks on page 1799

To edit an xref in a separate window

1. Click View tab ➤ Palettes panel ➤ External References Palette. 
2. In the External References palette, select the xref that you want to edit.
3. Right-click, and click Open.
4. The source file for the selected xrefs opens in a new window, where you can edit, save, and close the drawing.

To edit a nested xref in a separate window

1. Select an xref that contains nested xrefs.
2. At the command prompt, enter **xopen** and press Enter.
3. In the Open Reference Files dialog box, select a nested xref from the reference tree. Press Ctrl to select multiple xrefs.
4. Click Open.

   The source file for the selected xrefs opens in a new window, where you can edit, save, and close the drawing.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the External References palette.

XOPEN

Opens a selected drawing reference (xref) in a new window.

Edit Selected Objects in Referenced Drawings and Blocks

You can modify external references and redefine block definitions from within the current drawing by using in-place reference editing. Both blocks and xrefs are considered references.

By editing the reference in place, you can modify the reference within the visual context of your current drawing.

Often, a drawing contains one or more xrefs as well as multiple block references. When working with block references, you can select a block, modify it, view and edit its properties, and update the block definition.

When working with xrefs, you can select the reference you want to work with, modify its objects, and save back the changes to the reference drawing. You can make minor changes without having to go back and forth between drawings.

TIP You can quickly copy one or more selected objects nested within a referenced drawing or a block into the currect drawing with the NCOPY command.

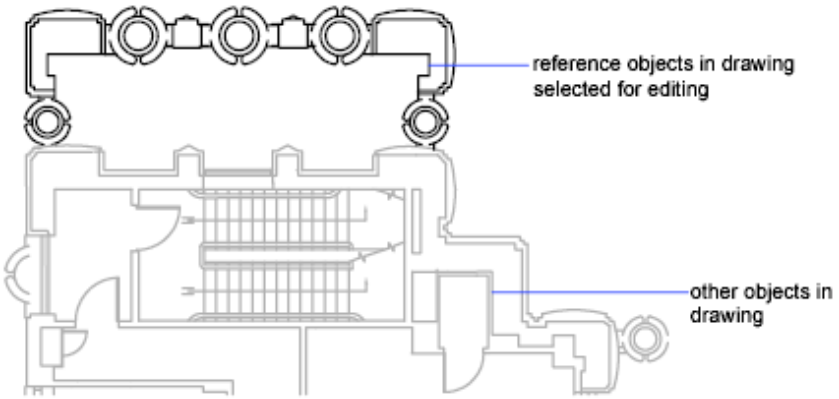
Understand the Working Set

The objects that you select from the selected xref or block are temporarily extracted and made available for editing in the current drawing. The set of extracted objects is called the *working set*, which can be modified and then saved back to update the xref or block definition.

Objects that make up the working set are visually distinct from other objects in the drawing. All objects in the current drawing, except objects in the working set, are faded.

Control the Fading of Objects

The XFADECTL system variable controls how objects are displayed while a reference is edited in place. The set of objects extracted from the reference are displayed normally. All other objects in the drawing, including objects in the current drawing and in any references not belonging to the working set, are faded. The value indicates the intensity of display for objects not in the working set. The larger the value is for XFADECTL, the more the objects are faded.



NOTE Objects outside the working set are not faded during in-place reference editing unless VSCURRENT is set to a value of 2D wireframe.

Use the Refedit Ribbon Contextual Tab

If you select nested objects to edit when the ribbon is active, the Edit Reference ribbon contextual tab is displayed. Using the buttons on the Edit Reference ribbon contextual tab, you can add objects to or remove objects from the working set, and you can discard or save back changes to the reference. The Edit Reference ribbon contextual tab is automatically dismissed after you save back or discard changes made to the working set.

Use the Refedit Toolbar

When the ribbon is not active, The Refedit toolbar is displayed and activated after you select which nested objects to edit. Using the buttons on the Refedit toolbar, you can add objects to or remove objects from the working set, and you can discard or save back changes to the reference. Unless it is docked, the Refedit toolbar is automatically dismissed after you save back or discard changes made to the working set.

NOTE If you plan to make *major* changes to a reference, open the reference drawing and edit directly within the file. Using in-place reference editing to make major changes can increase the size of your current drawing file significantly during the in-place reference editing session.

See also:

- Edit a Referenced Drawing in a Separate Window on page 1797

- Edit Referenced Drawings and Blocks with Nesting, OLE, or Attributes on page 1806

To edit an xref or block reference in place

1. Click Insert tab ➤ Reference panel ➤ Edit Reference. 
2. From within the current drawing, select the reference that you would like to edit.

   If the object you select in the reference belongs to any nested references, all the references available for selection are displayed in the Reference Edit dialog box.
3. In the Reference Edit dialog box, select the specific reference that you want to edit.

   The reference file is locked to prevent multiple users from opening the file at once. You cannot edit a reference in place if the drawing file is being used by another user.
4. Click OK.
5. Select the objects you would like to edit in the reference and press Enter.

   The objects that you select become the working set. By default, all other objects are locked and faded.
6. Edit the objects in the working set. Click Save Back Changes to Reference.

   The objects in the working set are saved to the reference and the xref or the block is updated.

Quick Reference

Commands

NCOPY

Copies objects that are contained in an xref, block, or DGN underlay.

REFCLOSE

Saves back or discards changes made during in-place editing of a reference, either an xref or a block definition.

REFEDIT

Edits an xref or a block definition directly within the current drawing.

REFSET

Adds or removes objects from a working set during in-place editing of a reference, either an xref or a block definition.

SHADEMODE

System Variables

BINDTYPE

Controls how xref names are handled when binding xrefs or editing xrefs in place.

REFEDITNAME

Displays the name of the reference being edited.

XEDIT

Controls whether the current drawing can be edited in-place when being referenced by another drawing.

XFADECTL

Controls the amount of fading within a reference being edited in place. This setting affects only the objects that are not being edited in the reference.

Use the Working Set to Edit Referenced Drawings and Blocks

To edit a referenced drawing from within the current drawing, you use the *working set* to identify objects that belong to the xref or block definition rather than the current drawing.

While editing a reference in place, you can add or remove objects from the working set. If you create a new object while editing a reference in place, it is almost always added to the working set automatically. Objects that are not in the working set are displayed as faded in the drawing.

If a new object is created because of changes made to objects outside the working set, the new object is not added to the working set. For example, your drawing contains two lines that are not a part of the working set. If you edit the lines by using FILLET, a new arc is created between the two lines. The arc is not added to the working set.

When a reference object is part of the working set, you can select the object for editing even if it is drawn on a locked layer in the reference file. You can unlock the object's layer and make changes to the object. Changes made to the object can be saved, but the layer state remains the same in the reference file, whether it is locked or unlocked.

An object that is removed from the working set is added to the host drawing and removed from the reference when changes are saved back. An object that is added to the working set is removed from the host drawing, and is restored to the reference when the changes are saved back.

Use the External Reference Tab on the Ribbon

If you select a reference to edit in-place when the ribbon is active, the Edit Reference ribbon contextual tab is displayed. The editing buttons on the contextual tab (Add to Working Set, Remove from Working Set, Discard Changes, and Save Changes) are active only during in-place reference editing. The contextual tab is dismissed automatically after changes made to the reference are saved back or discarded.

When the ribbon is not active, the Refedit toolbar is displayed when a reference is being edited in place. The name of the selected reference is displayed in the toolbar. The editing buttons on the toolbar (Add to Working Set, Remove from Working Set, Close Reference, and Save Reference Edits) are active only during in-place reference editing. The Edit Block or Xref button is active whenever the toolbar is initiated and a reference editing session is not already in progress within the current drawing. The Refedit toolbar is dismissed automatically after changes made to the reference are saved back or discarded.



To add objects to the working set

1. Click Tools menu ➤ Xref And Block In-Place Editing ➤ Add To Working Set.
2. Select the objects you want to add. You can also set PICKFIRST to 1 and create a selection set before using the Add option.

   REFSET can be used only with objects in the space (paper space or model space) in which REFEDIT is initiated.

To remove objects from the working set

1. Click Tools menu ➤ Xref And Block In-Place Editing ➤ Remove from Working Set.
2. Select the objects you want to remove. You can also set PICKFIRST to 1 and create a selection set before using the Remove option.

   REFSET can be used only with objects in the space (paper space or model space) in which REFEDIT is initiated.

Quick Reference

Commands

REFCLOSE

Saves back or discards changes made during in-place editing of a reference, either an xref or a block definition.

REFEDIT

Edits an xref or a block definition directly within the current drawing.

REFSET

Adds or removes objects from a working set during in-place editing of a reference, either an xref or a block definition.

System Variables

XEDIT

Controls whether the current drawing can be edited in-place when being referenced by another drawing.

Save Back Edited Referenced Drawings and Blocks

While editing a referenced drawing or a block definition in place, you can save back or discard changes.

While editing a block reference in place, you either can *save back* or discard changes made to the reference. If you save back changes to a reference, the drawing is regenerated.

When the changes are saved back, the block definition is redefined and all instances of the block are regenerated to reflect the changes. If you choose to

discard the changes, the working set is deleted and the block reference returns to its original state.

Similarly, while editing an xref in place, you can save back or discard changes. Objects in the working set that inherit properties not originally defined in the xref retain those new properties. For example, an xref contains layers A, B, and C, and the drawing that references it contains layer D. If new objects are drawn on layer D during in-place reference editing and changes are saved back to the reference, layer D is copied to the xref drawing.

If you remove objects from the working set and save changes, the objects are removed from the reference and added to the current drawing. Any changes you make to objects in the current drawing (not in the xref or block) are not discarded. If you delete any object that is not in the working set, the object is not restored even if you choose to discard changes. You can return the drawing to its original state by using UNDO. If you make unwanted changes to an xref and use REFCLOSE to save back the changes, you must use UNDO to undo any changes made during the reference editing session. After you have undone any unwanted changes, use REFCLOSE to save changes to restore the xref file to its original state.

WARNING While editing a reference in place, if you delete an object that is not in the working set, the object is not restored if you discard changes at the closing of the reference editing session.

Objects in the current drawing that inherit properties defined by the xref retain those new properties. Properties taken from the xref drawing are bound to the current drawing. The xref layer named SITE, for example, appears in the current drawing as $#$SITE when assigned to an object not in the working set.

If BINDTYPE is set to 0, a prefix of $#$ is added to the reference name in the current drawing. If BINDTYPE is set to 1, reference names remain unchanged in the current drawing, similar to names of inserted objects.

NOTE When you edit and save an xref in place, the original drawing preview is no longer available unless you open and save the referenced drawing.

To save back changes from edited xrefs and blocks

- Click Tools menu ➤ X ref And Block In-Place Editing ➤ Save Reference Edits. 

NOTE Alternatively, click the Save Back Changes to Reference button on either the Edit Reference ribbon contextual tab or the Refedit toolbar.

To discard all changes from edited xrefs and blocks

- On either the Edit Reference ribbon contextual tab or the Refedit toolbar, click the Discard Changes to Reference button.

Quick Reference

Commands

REFCLOSE

Saves back or discards changes made during in-place editing of a reference, either an xref or a block definition.

XOPEN

Opens a selected drawing reference (xref) in a new window.

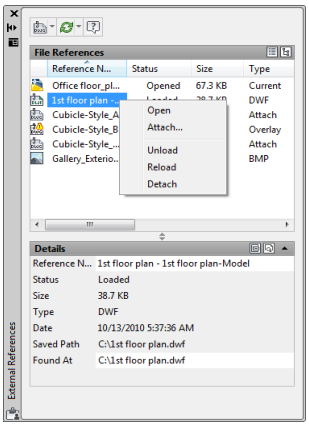
System Variables

BINDTYPE

Controls how xref names are handled when binding xrefs or editing xrefs in place.

Edit Referenced Drawings and Blocks with Nesting, OLE, or Attributes

If the reference you select for editing has attached xrefs or block definitions, the reference and its nested references are displayed and available for selection in the Reference Edit dialog box. Nested references are displayed only if the object chosen for selection is part of a nested reference. Only one reference at a time can be selected for editing. If you are editing a reference that contains OLE objects, the OLE objects are displayed but cannot be selected for editing.



If a block reference with attributes is selected for editing, you can choose to display the attribute definitions in the reference and make them available for editing. The attributes are made invisible and the attribute definitions are available for editing along with the selected reference geometry. When changes are saved back to the block reference, the attributes of the original reference remain unchanged. The new or altered attribute definitions only affect subsequent insertions of the block; the attributes in existing block instances are not affected.

See also:

- Edit a Referenced Drawing in a Separate Window on page 1797
- Edit Selected Objects in Referenced Drawings and Blocks on page 1799

Quick Reference

Commands

EXTERNALREFERENCES

Opens the External References palette.

REFEDIT

Edits an xref or a block definition directly within the current drawing.

REFSET

Adds or removes objects from a working set during in-place editing of a reference, either an xref or a block definition.

Resolve Referenced Drawing Errors

If a referenced drawing cannot be loaded when you open a drawing, an error message is displayed.

Resolve Missing External References

If a referenced drawing cannot be located when you open a drawing, several options available to you.

The program stores the folder path of the referenced drawing. Each time you open or plot the drawing, or use the Reload option in the External References palette to update the xref, the program checks the folder path to determine the name and location of the referenced drawing file.

IIf the name or location of the drawing file has changed, the program cannot locate or reload the xref, and it displays an error message that displays the folder path and name of the missing drawing file.

In the drawing, at each insertion of the missing xref, the program displays text that displays the folder path of the missing xref. You can use the XREF Path option to update or correct the path.

Along with error messages being displayed at the Command prompt, a task dialog box might be displayed that allows you to ignore all missing xrefs or update their folder locations. You can use the External References palette to update the locations of the unresolved references.

To avoid these errors make sure that when you transfer or distribute drawing files that have xrefs attached, you also include all the referenced files.

Change Nested Xref Paths

When a drawing is opened and a nested xref is loaded, the program attempts to find the xref in the original xref path first. If the xref is not found, the following search is initiated in the order shown:

- Current folder of the host drawing
- Project search paths defined on the Files tab in the Options dialog box and in the PROJECTNAME system variable
- Support search paths defined on the Files tab in the Options dialog box
- Start In folder specified in the Microsoft® Windows® application shortcut

This search order helps ensure that revisions made to the xref are reflected in the current drawing, and also makes it possible for the xref to be found if its folder path has changed.

See also:

- Update Referenced Drawing Attachments on page 1788

To change an xref path



1. Click View tab ➤ Palettes panel ➤ External References Palette.
2. In the External References palette, select the external reference.
3. Under Found At, do one of the following:
   - Edit the xref path directly.
   - Click the [...] button and browse to the xref in its new path.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the External References palette.

Resolve Circular External References

If a referenced drawing contains a sequence of nested references that refers back to itself, an error message is displayed.

A drawing that contains a sequence of nested references that refers back to itself is considered a circular reference. For example, if drawing A attaches drawing B, which attaches drawing C, which attaches drawing A, the reference sequence A>B>C>A is a circular reference.

If the program detects a circular reference while attaching an xref, a warning is displayed asking you if you want to continue. If you respond with yes, the program reads in the xref and any nested xrefs to the point where it detects the circularity. If you respond with no, the process is halted and the xref is not attached.

If a circular reference is encountered while loading a drawing, an error message is displayed and the circular reference for the current session is broken. For example, if you have the circular reference A>B>C>A, and you open *a.dwg*, the program detects and breaks the circularity between *c.dwg* and *a.dwg*. The following error message is displayed:

Breaking circular reference from C to current drawing.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the External References palette.

Resolve Name Conflicts in External References

When you attach an xref, the names of its blocks, dimension styles, layers, linetypes, and text styles are differentiated from those in the current drawing.

A typical xref definition includes objects, such as lines or arcs. It also includes xref-dependent definitions of blocks, dimension styles, layers, linetypes, and text styles. When you attach an xref, the program differentiates the names of these xref-dependent named objects from those in the current drawing by preceding their names with the name of the referenced drawing and a vertical bar character (|). For example, in the Layer Properties Manager, the xref-dependent named object that is a layer named STEEL in a referenced drawing called *stair.dwg* is listed as STAIR|STEEL.

When you attach an xref, the definitions of its dependent named objects are not added to your drawing permanently. Instead, these definitions are loaded from the referenced drawing file each time you reload it.

Bind Xref-Dependent Definitions

An xref-dependent named object's definition can change if the referenced drawing file is modified. For example, a layer name from a referenced drawing can change if the referenced drawing is modified. The layer name can even disappear if it is purged from the referenced drawing. This is why the program does not allow you to use an xref-dependent layer or other named object directly. For example, you cannot insert an xref-dependent block or make an xref-dependent layer the current layer and begin creating new objects on it.

To avoid the restrictions on xref-dependent named objects, you can bind them to your current drawing. Binding makes the xref-dependent named objects that you select become a permanent part of your current drawing.

When xref-dependent named objects are merged into a drawing through binding, you can use them the same way you use the drawing's own named objects. After you bind an xref-dependent named object, the vertical bar character (|) is removed from the name and replaced with two dollar signs ($$) separated by a number (usually zero): for example, the referenced layer, STAIR|STEEL, becomes STAIR0STEEL. You can then use the RENAME command to change STAIR0STEEL to STEEL.

If you specify a layer whose associated linetype is not CONTINUOUS, the referenced linetype is also bound. If you apply XBIND to a block, all named objects that are referenced by the objects in the block are also bound. If the block contains a reference to an xref, that xref and all of its dependent definitions are bound.

To bind xref-dependent named objects to the current draw ing

1. Click Modify ➤ Object ➤ External Reference ➤ Bind.
2. In the Xbind dialog box, click the plus (+) sign next to an external reference.

   The five types of named object definitions (Block, Dimstyle, Layer, Linetype, and Textstyle) are listed.
3. Click the plus (+) sign of one of the definition types.

   The names of the definition table entries are listed.
4. Select a named object definition. Click Add.

   The named object definition is listed under Definitions to Bind.
5. If necessary, repeat steps 3 and 4.
6. Click OK.

To change the names of layers, dimension styles, and other named objects

1. Click Format ➤ Rename. 
2. In the Rename dialog box, select the named object type and then the item you want to rename.
3. Enter the new name in the Rename To box below the old name.

4 Click Rename To. Click OK.

Quick Reference

Commands

RENAME

Changes the names assigned to items such as layers and dimension styles.

XBIND

Binds one or more definitions of named objects in an xref to the current drawing.

System Variables

BINDTYPE

Controls how xref names are handled when binding xrefs or editing xrefs in place.

Track External Reference Operations (Log File)

You can maintain a record of actions while attaching, detaching, and reloading xrefs, and while loading a drawing containing xrefs.

This log is maintained only if the XREFCTL system variable is set to 1. The default setting is 0.

The log file is an ordinary ASCII text file with the same name as the current drawing and the file extension *.xlg*. If you load a drawing with the file name *sample.dwg*, for example, the program searches for a log file named *sample.xlg* in the current folder. If the file does not exist, a new file is created with that name.

Once a log file has been created for a drawing, the program continues to append information to it. The program writes a title section to the log file each time the file is opened. If the log file becomes too large, you can delete it.

Example: A Sample Title Section from an Xref Log File

This title section contains the name of the current drawing, the date and time, and the operation being performed.

```
==============================
Drawing: detail
Date/Time: 09/28/99 10:45:20
Operation: Attach Xref
==============================
```

During a detaching or reloading operation, the program includes the nesting level of all affected xrefs immediately following the title section. To see a reference tree for a set of xrefs in your current drawing, use Detach or Reload and check the resulting entries in the log file.

Example: A Sample Log File Entry Showing Nested Xrefs

In the following example, the xref ENTRY_DR contains two nested xrefs: HARDWARE and PANELS. The xrefs HARDWARE and PANELS also each contain two xrefs.

```
==============================
Drawing: detail
Date/Time: 10/05/99 15:47:39
Operation: Reload Xref
==============================
Reference tree for ENTRY_DR:
ENTRY_DR Xref
-HARDWARE Xref
--LOCKSET Xref
--HINGES Xref
-PANELS Xref
--UPPER Xref
--LOWER Xref
```

The program writes an entry in the log file for each xref-dependent named object temporarily added to the current drawing and for any errors that occur. Most error messages are written both to the screen and to the log file.

Example: A Sample Log File That Shows the Results of Attaching an Xref

The following example shows a partial listing of the log file entries generated when the external reference STAIR is attached to the working drawing *test.dwg*.

The log file lists the definition (symbol) table affected and the name of the definition added, along with a status message.

```
==============================
Drawing: test
Date/Time: 12/18/99 14:06:34
Operation: Attach Xref
==============================
Attach Xref STAIR: \ACAD\DWGS\STAIR.dwg
 Searching in ACAD search path
 Update block symbol table:
 Appending symbol: STAIR|BOLT
 Appending symbol: STAIR|BOLT-HALF
 ...
 block update complete.
 Update Ltype symbol table:
 Appending symbol: STAIR|DASHED
 Appending symbol: STAIR|CENTER
 Appending symbol: STAIR|PHANTOM
 Ltype update complete.
 Update Layer symbol table:
 Appending symbol: STAIR|STEEL-HIDDEN
 Appending symbol: STAIR|OAK
 ...
 Layer update complete.
STAIR loaded.
```

To use the xref log file

1. At the Command prompt, enter **xrefctl**.
2. Enter **1** to turn logging on or **0** to turn logging off.
3. Press Enter.
   Logging is off by default.

Quick Reference

Commands

XREF

Starts the EXTERNALREFERENCES command.

System Variables

XREFCTL

Controls whether external reference log (XLG) files are created.

Increase Performance with Large Referenced Drawings

There are several features that can improve performance when dealing with large referenced drawings.

Overview of Demand Loading

The program uses *demand loading* and saving drawings with internal indexes to increase performance with large referenced drawings that have been clipped, or that have many objects on frozen layers. With demand loading, only the data from the reference drawing that is necessary to regenerate the current drawing is loaded into memory. In other words, referenced data is read in "on demand."

Demand loading works in conjunction with the INDEXCTL, XLOADCTL, and XLOADPATH system variables.

Quick Reference

Commands

XREF

Starts the EXTERNALREFERENCES command.

System Variables

INDEXCTL

Controls whether layer and spatial indexes are created and saved in drawing files.

XLOADCTL

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

XLOADPATH

Creates a path for storing temporary copies of demand-loaded xref files.

Unload Xrefs in Large Drawings

When a referenced drawing (xref) is unloaded from the current drawing, the drawing opens much faster and uses less memory.

The xref definition is unloaded from the drawing file, but the internal pointer to the referenced drawing remains. The xref is not displayed, and nongraphical object information does not appear in the drawing. However, you can restore all the information by reloading the xref. If XLOADCTL (demand loading) is set to 1, unloading the drawing unlocks the original file.

You should unload a reference file if it is not needed in the current drawing session but may be used later for plotting. You can maintain a working list of unloaded xrefs in the drawing file that you can load as needed.

To unload an xref

1. Click View tab ➤ Palettes panel ➤ External References Palette. 
2. In the External References palette, select the reference name you want to unload.
3. Right-click, and click Unload.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the External References palette.

System Variables

XLOADCTL

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

Work with Demand Loading in Large Drawings

With demand loading, only the data from the referenced drawing that is necessary to regenerate the current drawing is loaded into memory.

To realize the maximum benefits of demand loading, you need to save the referenced drawings with layer and spatial indexes. The performance benefits of demand loading are most noticeable when you do one of the following:

- Clip the xref with the program to display a small fraction of it. A spatial index is saved in the externally referenced drawing.
- Freeze several layers of the xref. The externally referenced drawing is saved with a layer index.

If demand loading is turned on, and you have clipped xrefs that were saved with spatial indexes, objects in the referenced drawing database that are contained within the clip volume comprise the majority of the objects read into the drawing. If the clip volume is modified, more objects are loaded as required from the reference drawing. Similarly, if you have xrefs with many layers frozen that were saved with layer indexes, only the objects on those thawed layers are read into the current drawing. If those xref-dependent layers are thawed, the program reads in that geometry from the reference drawing as required.

When demand loading is turned on, the program places a lock on all reference drawings so that it can read in any geometry it needs to on demand. Other users can open those reference drawings, but they cannot save changes to them. If you want other users to be able to modify an xref that is being demand loaded into another drawing, use demand loading with the Copy option.

If you turn on demand loading with the Enable with Copy option, the program makes a temporary copy of the referenced drawing and demand loads the temporary file. You can then demand load the xref while allowing the original reference drawing to be available for modification. When you turn off demand loading, the program reads in the entire reference drawing regardless of layer visibility or clip instances.

Layer and spatial indexes were added in AutoCAD Release 14 and AutoCAD LT 97. If you externally reference a drawing saved in a release previous to this, you do not see the same performance benefit as drawings saved with the indexes. For maximum performance, use demand loading with referenced drawings saved with layer and spatial indexes turned on in AutoCAD Release 14, AutoCAD LT 97, or more recent versions.

To turn on demand loading

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Open and Save tab, under External References, select Disabled, Enabled, or Enabled with Copy.

3 Click OK.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the External References palette.

System Variables

INDEXCTL

Controls whether layer and spatial indexes are created and saved in drawing files.

XLOADCTL

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

Work with Layer and Spatial Indexes

To receive the maximum benefit of demand loading, it is recommended that you save any drawings that are used as xrefs with layer and spatial indexes.

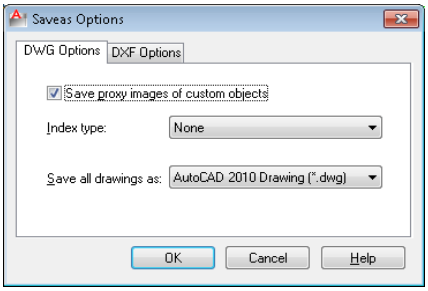
A layer index is a list showing which objects are on which layers. This list is used when the program is referencing the drawing in conjunction with demand loading to determine which objects need to be read in and displayed. Objects on frozen layers in a referenced drawing are not read in if the referenced drawing has a layer index and is being demand loaded.

The spatial index organizes objects based on their location in 3D space. This organization is used to efficiently determine which objects need to be read in when the drawing is being demand loaded and clipped as an xref. If demand loading is turned on, and the drawing is attached as an xref and clipped, the program uses the spatial index in the externally referenced drawing to determine which objects lie within the clip boundary. The program then reads only those objects into the current session.

Spatial and layer indexes are best used in drawings that will be used as xrefs in other drawings where demand loading is enabled. Drawings that are not going to be used as xrefs or partially opened will not benefit from layer and spatial indexing or demand loading.

To save a drawing with layer and spatial indexes

1. Click File menu ➤ Save As.
2. In the Save Drawing As dialog box, click Tools ➤ Options.



3. In the Saveas Options dialog box, under Index Type, select Layer, Spatial, or Layer & Spatial. Click OK.

NOTE If you are saving a partially open drawing that does not already contain layer and spatial indexes, this option is disabled.

4. Click Save.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the External References palette.

System Variables

INDEXCTL

Controls whether layer and spatial indexes are created and saved in drawing files.

TREEDEPTH

Specifies the maximum depth, that is, the number of times the tree-structured spatial index can divide into branches.

TREEMAX

Limits memory consumption during drawing regeneration by limiting the number of nodes in the spatial index (oct-tree).

XLOADCTL

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

Set Paths for Temporary Xref File Copies

When you turn on demand loading with copy, you can control where copies of externally referenced drawings are to be placed.

When you turn on demand loading with copy, the XLOADPATH system variable can be used to indicate the path where copies of externally referenced drawings are to be placed. The path you specify remains in effect for all drawing sessions until you indicate a different path. If no value for XLOADPATH is specified, the temporary file copies are placed in the standard folder for temporary files.

If you find that referencing drawings over a network is slow, it is recommended that you set XLOADPATH to reference a local folder, and set XLOADCTL to 2 so that the externally referenced files are demand loaded from your local machine. Conversely, to minimize the number of temporary files created by multiple users referencing the same drawing, those users can set XLOADPATH to point to a common folder. In this manner, multiple sessions of the program can share the same temporary copies of reference drawings.

You can set XLOADPATH in the Options dialog box and indicate the path where copies of externally referenced files are to be placed.

To set the file path for xref copies

1. Click Application menu ➤ Options.
2. In the Options dialog box, Files tab, select the *Temporary External Reference File Location* folder.

   The expanded tree view displays the path where copies of xref files are placed. If no path is specified, the temporary copies are placed in the location specified by Temporary Drawing File Location.
3. Edit the path by selecting it and entering a new path.
4. Click OK (or Apply) to set the path.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the External References palette.

OPTIONS

Customizes the program settings.

System Variables

XLOADPATH

Creates a path for storing temporary copies of demand-loaded xref files.

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Link and Embed Data (OLE) 36

Object linking and embedding (OLE) is a Windows feature that combines data from different applications into one document. For example, you can create an Adobe PageMaker layout that contains an AutoCAD drawing, or you can create an AutoCAD drawing that contains all or part of a Microsoft Excel spreadsheet.

Overview of Object Linking and Embedding

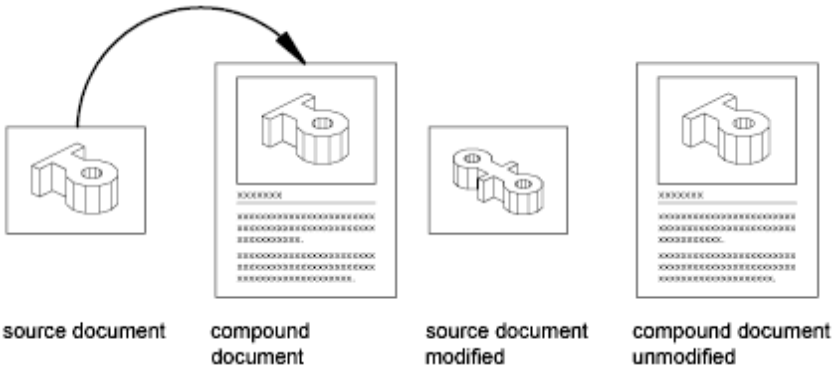
Object linking and embedding is a way to use information from one application in another application. To use OLE, you need both source and destination applications that support OLE.

Both linking and embedding insert information from one document into another document. Also, both linked and embedded OLE objects can be edited from within the destination application. However, linking and embedding store information differently.

The relationship between embedding and linking is similar to that between inserting a block and creating an external reference.

Embed Objects

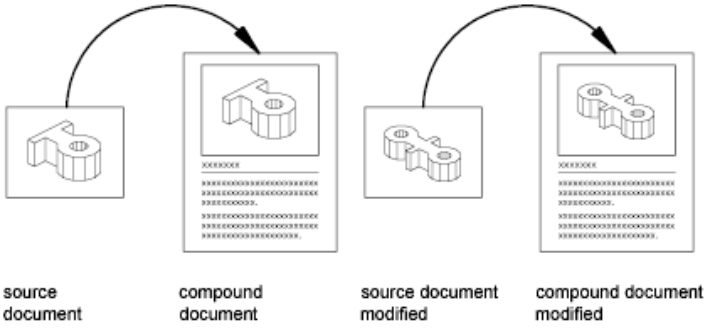
An embedded OLE object is a copy of information from another document. When you embed objects, there is no link to the source document and any changes made to the source document are not reflected in destination documents. Embed objects if you want to be able to use the application that created them for editing, but you do not want the OLE object to be updated when you edit information in the source document.



Link Objects

A linked object is a reference to information in another document. Link objects when you want to use the same information in more than one document. Then, if you change the original information, you need to update only the links in order to update the document containing the OLE objects. You can also set links to be updated automatically.

When you link a drawing, you need to maintain access to the source application and the linked document. If you rename or move either of them, you may need to reestablish the link.



Control the Plot Quality of OLE Objects

OLE objects are treated as raster objects when a raster plotter is used. Because large, high-resolution, color-rich rasters can be expensive to plot, you can set the OLEQUALITY system variable to control how each OLE object is plotted. The default setting, Automatically Select, assigns a plot-quality level based on the type of object. The higher the plot-quality setting, the more time and memory are used to plot.

You can also adjust OLE plot quality in the Plotter Configuration Editor. The Graphics option displays a Raster Graphics dialog box with a slider that controls OLE plot quality.

NOTE Nested OLE objects may cause problems. For example, an Excel spreadsheet embedded in a Word document may fail to plot. Also, an OLE object that is not in the current view plane is not plotted, but the frame is plotted based on the setting of the OLEFRAME system variable.

See also:

- Cut, Copy, and Paste with the Clipboard on page 772

To set the plot quality for OLE objects

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Plot and Publish tab, in the OLE Plot Quality list, select one of the following settings:
   - **Monochrome.** For example, spreadsheets
   - **Low Graphics.** For example, color text and pie charts
   - **High Graphics.** For example, photographs
   - **Automatically Select.** Plot-quality setting assigned based on the type of file
3. Click Apply to continue setting options, or click OK to close the dialog box.

Quick Reference

Commands

COPYLINK

Copies the current view to the Clipboard for linking to other OLE applications.

INSERTOBJ

Inserts a linked or embedded object.

OLELINKS

Updates, changes, and cancels existing OLE links.

System Variables

OLEHIDE

Controls the display and plotting of OLE objects.

Import OLE Objects

You can import information into a drawing from other applications that support OLE.

Overview of Importing OLE Objects into Drawings

Use one of the following methods to insert information from another application as an OLE object:

- Copy or cut information from an existing file and paste it into the drawing.
- Import an existing file created in another application.
- Open another application from within the drawing and create the information that you want to use.

When you insert the information, you specify an insertion point.

By default the OLE object is displayed with a frame that is not plotted. OLE objects are opaque and are plotted as opaque; they hide objects in back of them. OLE objects support draw order. You control the display of OLE objects in two ways:

- Set the OLEHIDE system variable to display or suppress the display of all OLE objects in paper space, model space, or both.
- Turn off or freeze a layer to suppress the display of OLE objects on that layer.

When OLE objects with text are printed, the text size approximates the text size in the source application.

NOTE OLE objects in drawings are not displayed or plotted in external references or block references.

Quick Reference

Commands

INSERTOBJ

Inserts a linked or embedded object.

OLELINKS

Updates, changes, and cancels existing OLE links.

OLESCALE

Controls the size, scale, and other properties of a selected OLE object.

OPTIONS

Customizes the program settings.

PASTESPEC

Pastes objects from the Clipboard into the current drawing and controls the format of the data.

System Variables

MSOLESCALE

Controls the size of an OLE object with text that is pasted into model space.

OLEHIDE

Controls the display and plotting of OLE objects.

Link OLE Objects in Drawings

When you link information from another document in a drawing, the information can be updated when it changes in the source document.

You can link information from a document created by another application to a drawing. For example, you might want to insert a schedule that will be automatically updated. You can also install multimedia icons that are activated when double-clicked.

Update Links

Links can be set to be updated either automatically or manually when information in the linked document changes. By default, links are updated automatically. Use OLELINKS to specify automatic or manual updating.

Reconnect Links

Because a link references the location of the linked document, you need to reconnect the link if the document changes location or is renamed.

Break Links

Breaking a link does not remove the inserted information from the drawing. Instead, it removes the connection to the linked document. You can break a link when you no longer need to update the information.

To link objects in a drawing

1. Start the source application and open a document.
2. Select the information you want to link and copy it to the Clipboard.
3. Open the drawing.
4. Click Home tab ➤ Clipboard panel ➤ Paste drop-down ➤ Paste Special.



5. In the Paste Special dialog box, click Paste Link.

   Paste Link pastes the contents of the Clipboard into the current drawing and creates a link to the file in the source application. If you click Paste, the Clipboard contents are embedded instead of linked.
6. In the As box, select the data format you want to use.
7. Click OK.

To link an entire file as an OLE object in a drawing

1. Open the drawing.
2. Click Insert tab ➤ Data panel ➤ OLE Object. 
3. In the Insert Object dialog box, select Create from File.
4. Select Link. Click Browse.
5. In the Browse dialog box, select the file you want to link. Click Open.
6. In the Insert Object dialog box, click OK.

   The entire file is linked to the drawing.

To update links manually

1. Click Edit menu ➤ OLE Links.
2. In the Links dialog box, select the links you want to update.
3. Click Update Now.
4. Click Close.

NOTE If you want to select the links from the drawing area rather than from a list in the Links dialog box, select the OLE objects before step 1.

To reconnect a link to an OLE object

1. In the drawing, select the OLE object whose link you want to change.
2. Click Edit menu ➤ OLE Links.
3. In the Links dialog box, click Change Source.
4. In the Change Source dialog box, search for the source file.
5. Select the source file. Click Open.
6. Click OK.

To break a link to an OLE object

1. Select the linked object whose link you want to break.
2. Click Edit menu ➤ OLE Links.
3. In the OLE Links dialog box, click Break Link.
4. Click Yes.
5. Click Close.

Quick Reference

Commands

INSERTOBJ

Inserts a linked or embedded object.

OLELINKS

Updates, changes, and cancels existing OLE links.

OLESCALE

Controls the size, scale, and other properties of a selected OLE object.

OPTIONS

Customizes the program settings.

OPTIONS

Customizes the program settings.

PASTESPEC

Pastes objects from the Clipboard into the current drawing and controls the format of the data.

System Variables

OLEHIDE

Controls the display and plotting of OLE objects.

Embed OLE Objects in Drawings

When you embed information from another document in a drawing, the information is not updated when it changes in the source document.

You can embed an object in a drawing by copying the object to the Clipboard and then pasting it into the drawing file. For example, you can embed a company logo created with another application into a drawing.

Drag Objects into a Drawing

You can drag selected data and graphics into a drawing from another application window. Both applications must be running and visible on your screen. The other application must support ActiveX to drag information between applications. Objects dragged into a drawing are embedded, not linked.

Dragging data is the same as cutting and pasting. The information is removed from one document and pasted into the other document. Holding down CTRL while dragging is the same as copying and pasting in that it creates a copy of the information, leaving the original document intact.

To embed an OLE object in a drawing

1. Open the document in the source application.
2. Copy the information you want to embed to the Clipboard.
3. Open the drawing.
4. Click Home tab ➤ Utilities panel ➤ Paste. 
5. Click OK.

To create an object in another application and embed it in a drawing

1. Open the drawing.
2. Click Blocks & References tab ➤ Data panel ➤ OLE Object. 
3. In the Insert Object dialog box, under Object Type, select Create New.
4. Under Object Type, select an application. Click OK.
   The source application opens.
5. Using the server application, create the information you want to insert. Save the document.
6. On the source application's File menu, click Exit and Return.
7. Close the source application.
   The OLE object is embedded in the drawing.

To specify a new height or width for an OLE object

1. Right-click an OLE object. Click Properties.
2. In the Properties palette, enter new values for Width or Height, or enter a percentage for Scale Width or Scale Height.

NOTE When Lock Aspect is set to Yes, if you change either height or width, the other automatically changes to maintain the current ratio between the two. For example, if you change height to 50 percent, width automatically changes to 50 percent. Set Lock Aspect to No if you want to change only height or only width.

3 Click OK to apply changes.

To scale text in an OLE object according to its font

1 Select an OLE object.

2 Right-click. Click Text Size.

3 In the OLE Text Size dialog box, select a font.

The OLE Text Font list contains all of the fonts that appear in the selected OLE object.

4 Select a point size.

The OLE Point Size list contains all of the sizes for the selected font.

5 In Text Height, enter a value in drawing units.

This value is the height for text in the selected font and point size. For example, if you select Arial and 10 points, and then enter **.5** in Text Height, all text in the selected OLE object that is currently 10-point Arial changes to 0.5 drawing units in height. All other text in the object changes size in relation to the selected font. The size of the OLE object adjusts to accommodate the new text sizes.

Quick Reference

Commands

INSERTOBJ

Inserts a linked or embedded object.

OLESCALE

Controls the size, scale, and other properties of a selected OLE object.

OPTIONS

Customizes the program settings.

OPTIONS

Customizes the program settings.

PASTECLIP

Pastes objects from the Clipboard into the current drawing.

PASTESPEC

Pastes objects from the Clipboard into the current drawing and controls the format of the data.

System Variables

OLEHIDE

Controls the display and plotting of OLE objects.

OLEQUALITY

Sets the default plot quality for OLE objects.

OLESTARTUP

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

Export OLE Objects from Drawings

You can link or embed a view of a drawing in another application that supports OLE.

Link Views to Other Documents

A drawing can be a source document linked to one or more documents in other applications. The COPYLINK command copies the view in the current viewport to the Clipboard. You can then paste the view into the destination document.

If you paste an unnamed view into a document, it is assigned a view name such as OLE1. If you then exit the drawing, you are prompted to save your changes to the newly named view. To establish the link and to save the view name, OLE1, you must save the drawing.

Embed Drawing Objects in Other Documents

You can select objects and embed them in documents created by other applications. Embedding places a copy of the selected objects in the destination document. If you use AutoCAD to edit the OLE object from within the destination document, the object is not updated in the original drawing.

To embed objects in another document

1. Click Edit menu ➤ Copy. 
2. In the drawing area, select the objects that you want to embed and then press ENTER.

   The selected objects are copied to the Clipboard.
3. Start the destination application, and open a new or existing document.
4. Paste the Clipboard contents into the document, following the destination application's instructions for embedding the Clipboard contents.

To link a view to another document

1. Save the drawing that you want to link, so that it has a drawing name.
2. If multiple viewports are displayed, select a viewport.
3. Enter **copylink** at the Command prompt.
4. Open a new or existing document in the destination application.
5. Paste the Clipboard contents into the document, following that application's procedures for inserting linked data.

   The inserted OLE object is displayed in the document and can be edited from AutoCAD through the destination application.

Quick Reference

Commands

COPYCLIP

Copies selected objects to the Clipboard.

COPYLINK

Copies the current view to the Clipboard for linking to other OLE applications.

CUTCLIP

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

INSERTOBJ

Inserts a linked or embedded object.

OLELINKS

Updates, changes, and cancels existing OLE links.

System Variables

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.

Edit OLE Objects in Drawings

You can edit a linked or embedded OLE object in a drawing by double-clicking the object to open the source application.

You can use any selection method to select OLE objects and then use most editing commands, the Properties palette, or grips to make changes. When you use grips to change the size of an OLE object, the shape of the object does not change if the aspect ratio is locked in the Properties palette. The following editing commands are not available for OLE objects: BREAK, CHAMFER, FILLET, and LENGTHEN.

In a 2D visual style, when an OLE object is rotated or is not in Plan view, the content of the OLE object is temporarily hidden and only the frame is displayed. The content is always shown in a 3D visual style.

In the Properties palette, the general properties listed for an OLE object apply to the frame.

Because grips are displayed on the frame, grip editing is not available if the frame is not displayed. To display the frame, change the setting of the OLEFRAME system variable.

Edit Information in OLE Objects

You can edit the information in linked or embedded OLE objects by double-clicking the object to open the source application.

Edit OLE Objects When AutoCAD Is the Source Application

The document that contains a linked drawing stores the drawing's file location. You can edit a linked drawing either from the destination application or in the source program. The program must be loaded or accessible on the system along with the document you are editing.

An AutoCAD drawing that is embedded in a document can be edited only from within the destination application. Double-click the OLE object to start the program. Editing the original drawing in the program has no effect on documents in which that drawing is embedded.

See also:

- Copy Objects on page 796

To edit a linked drawing from within the destination application

1. Open the document that contains the linked drawing (for example, a Microsoft Word file).
2. Double-click the linked drawing.

   The drawing opens.
3. Modify the drawing as necessary.
4. To save the changes to the drawing, click File menu ➤ Save.
5. To return to the destination application, click File menu ➤ Exit.

   The drawing is changed in all documents that have links to it.

   How the link is updated depends on the destination application. Some applications support automatic updating of links; others require manual updating of links.

To edit a linked drawing in the source application

1. Start the program and open the linked drawing.
2. Modify the drawing and view as necessary.
3. To save the changes to the drawing, click File menu ➤ Save.
4. Update the link in the destination document if necessary.

   The drawing is changed in all documents that have links to it.

How the link is updated depends on the destination application. Some applications support automatic updating of links; others require manual updating of links.

To edit embedded objects

1. Open the document that contains the embedded AutoCAD objects (for example, a Microsoft Word file).
2. Double-click the embedded objects to start the program and display the objects.
3. Modify the objects as necessary.
4. To save changes to the embedded objects, click File menu ➤ Update.
5. To return to the destination application, click File menu ➤ Exit.

To restore an OLE object to its original size and shape

1. Select the OLE object.
2. Right-click.Click OLE ➤ Reset.
   The Reset option is also available in the Text Size dialog box.

To control the display of OLE objects

1. At the Command prompt, enter **olehide.**
2. Enter one of the following values:
   - **0** Displays OLE objects in both paper space and model space.
   - **1** Displays OLE objects in paper space only.
   - **2** Displays OLE objects in model space only.
   - **3** Does not display OLE objects.

To turn the display of the frames of OLE objects on or off

1. At the Command prompt, enter **oleframe.**
2. Enter one of the following values:
   - **0** Frame is not displayed and not plotted.
   - **1** Frame is displayed and is plotted.

- **2** Frame is displayed but is not plotted.

The frame must be displayed in order for grips to be visible.

Quick Reference

Commands

COPYCLIP

Copies selected objects to the Clipboard.

CUTCLIP

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

ERASE

Removes objects from a drawing.

PASTECLIP

Pastes objects from the Clipboard into the current drawing.

U

Reverses the most recent operation.

System Variables

OLEFRAME

Controls whether a frame is displayed and plotted on all OLE objects in the drawing.

OLEHIDE

Controls the display and plotting of OLE objects.

Work with Data in Other Formats

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You can work with many different types of files, including files created with other applications and files created in earlier releases of the program. You can also specify search paths for drawing and support files.

Convert Drawing File Formats

You can convert AutoCAD-based drawing files to other AutoCAD drawing formats.

Choose the file format, conversion options, and the type of output you want.

With DWGCONVERT, you can convert AutoCAD DWG files between the following data formats:

- AutoCAD 2010/LT 2010
- AutoCAD 2007/LT 2007
- AutoCAD 2004/LT 2004
- AutoCAD 2000/LT 2000
- AutoCAD R14, AutoCAD LT 98, AutoCAD LT 97

NOTE The releases of AutoCAD-based products between the drawing file format changes listed above are based on the previous format.

When converting drawing files, select them within the dialog box without opening them. Files can be converted one at a time, or in batches; either

overwriting the originals or packaged together in a file folder, compressed in a self-extracting EXE file, or compressed in a ZIP file in a different location.

Specify the Output of the Conversion

The conversion setup controls the output for the converted drawing files. In the Modify Conversion Setup dialog box, under Conversion Package Type, choose from the following:

- Replace (overwrite) the existing files
- Create the drawing files in a specified folder
- Combine the drawing files in a self-extracting executable file
- Combine the drawing files in a ZIP file

The conversion package type you choose controls access to several other options in the dialog box.

To specify conversion setup options

1. Click Application button ➤ Save As ➤ DWG Convert.
2. In the DWG Convert dialog box, click Conversion Setups.
3. In the Conversion Setups dialog box, do *one* of the following:
   - Click a previously saved conversion setup from the list and click Close.
   - Click New to open the New Conversion Setup dialog box. After you enter a new name, click Continue to return to the Modify Conversion Setup dialog box.
   - Click a previously saved conversion setup and then click Modify. The Modify Conversion Setup dialog box is displayed.
4. (Optional) In the Modify Conversion Setup dialog box, specify all options that you want to change and click OK.
5. Click Close.

To convert an AutoCAD-based drawing to a different release format

1. Click Application button ➤ Save As ➤ DWG Convert.
2. In the DWG Convert dialog box, near the bottom, click the Add File button.

3 In the Select File dialog box, navigate to the folder of the drawing file that you want to convert.

4 Click the drawing file name and click Open.

5 (Recommended) In the DWG Convert dialog box, click Conversion Setups. In the Conversion Setups dialog box, create a new conversion setup, modify an existing conversion setup, or select a previously created conversion setup. Click Close.

6 In the DWG Convert dialog box, click Convert.

7 If the conversion setup specifies an EXE or ZIP file, navigate to and double-click the desired destination folder to select it. If desired, enter a new name for the file.

8 Click Save.

Quick Reference

DWGCONVERT

Converts drawing format version for selected drawing files.

Understand the Effect of Format Changes

You can convert a drawing to a format compatible with previous versions of the AutoCAD. However, you should be aware of several limitations.

This process creates a drawing with information specific to the current release stripped out or converted to another object type. When the drawing is converted, a log file lists changed or lost information.

Because converting a drawing to an earlier release format may cause some data loss, it is recommended to assign a different file name to avoid overwriting the current drawing file.

Dynamic Constraints

Dynamic contraints created in AutoCAD 2010-based products do not display in DWG TrueView.

Associativity in Dimensions

Associative dimensions created in AutoCAD 2002-based products or later generally maintain their associativity when converted to a previous release and then reopened in the current release of an AutoCAD-based product.

Visual Fidelity of Dimensions

The following dimension enhancements do not lose visual fidelity in previous releases if they are not edited:

- Dimension breaks
- Jogged linear dimensions
- Inspection dimensions
- Angular dimensions that are dimensioned using the quadrant option
- Arc extension lines for radial and diameter dimensions

Visual Fidelity for Annotative Objects

You can specify that *annotative objects* maintain visual fidelity when they are viewed in AutoCAD 2007-based products and earlier releases in the Modify Conversion Setup dialog box.

Annotative objects may have multiple scale representations. When visual fidelity is on, annotative objects are decomposed and scale representations are saved to separate layers. These layers are named based on their original layer and appended with a number.

When visual fidelity for annotative objects is not selected, a single model space representation is displayed on the Model tab. Depending on a setting when the drawing was saved, additional annotation objects might be displayed on the Model tab, and additional objects might be displayed in paper space viewports at different sizes than in AutoCAD 2008-based products and later releases.

Annotative Blocks

In an AutoCAD 2008-based drawing, when an annotative block does not have its paper orientation set to match the layout, and the block contains multiline attributes that are based on a text style that is not set to match the orientation of the layout, the attributes might shift positions when the drawing is opened in an AutoCAD 2007-based product or an earlier release.

Layer Property Overrides

If you convert an AutoCAD 2008-based drawing containing layer property overrides, the overrides are not visible in previous releases. The property override settings are retained when the drawing is saved in a previous release, and are visible again when the drawing is opened in an AutoCAD 2008-based product.

For previous releases of AutoCAD-based products, if the VISRETAIN system variable was set to 0 when the drawing was saved, xref layers containing viewport property overrides are not retained when opened in DWG TrueView.

If you open an AutoCAD 2008-based drawing in a previous release, property overrides might display in a thumbnail image. When the drawing is saved with a layout tab, and then opened in the previous release, those property overrides do not display.

Multiline Text Objects

Some of the paragraph spacing and paragraph line spacing options are not supported when an AutoCAD 2008 multiline text object is opened in AutoCAD 2007 and earlier releases.

The following multiline text formatting features have *no* visual fidelity in previous releases:

- Paragraphs with justified alignment
- Paragraphs with distributed alignment
- Fields that wrap across columns
- Fields that wrap across lines that have new paragraph alignments
- Paragraphs with non-default alignments in multiline text without left object-level justification

The following multiline text formatting features have *some* visual fidelity in previous releases (when it's possible to add white spaces or replace text with white spaces):

- Paragraphs with non-default alignments (other than justified or distributed) in multiline text that has left object-level justification
- Paragraphs with tabs using tab alignments (center, right, or decimal alignment applied)
- Paragraphs with line spacing that can be approximated with "tall" spaces

Multiline text with new formatting that is edited and saved in previous releases loses the new formatting when reopened in AutoCAD 2008-based products.

Tables and Data Extraction

Editing AutoCAD 2008 tables in previous releases removes AutoCAD 2008 table formatting. Also, AutoCAD 2008 table cells with long block and text strings may extend outside of cell borders when opened in previous releases.

For tables that were created with the Data Extraction wizard in AutoCAD 2008-based products, you can't edit or update the extracted data in previous releases.

Materials (Rendering)

AutoCAD 2008-based products introduced new procedural maps for materials and luminance. Drawings created earlier than AutoCAD 2007 that contain materials need to be converted. AutoCAD 2008 can automatically convert old materials to the AutoCAD 2008 format. However, there are some fidelity issues when saving back to previous versions of AutoCAD.

- Procedural maps Checker, Noise, Speckle, Tiles, and Waves are not available in AutoCAD 2007 and previous releases. They do not appear when an AutoCAD 2008 drawing is opened in AutoCAD 2007. Marble and Wood are available but the material might show some variation.
- The behavior of the Opacity map channel under the Realistic material type has been corrected in AutoCAD 2008. In AutoCAD 2008, the Opacity map channel behaves like a transparency map, where white is transparent and black is opaque (this was how it behaved prior to AutoCAD 2007). In AutoCAD 2007, white represented opaque areas and black represented transparent areas.
- The spherical map rotation is different in AutoCAD 2008-based products. When you open a drawing prior to AutoCAD 2008, the seam for the map is towards the +Y axis. In AutoCAD 2008, it is towards the +X axis.

Lighting (Rendering)

AutoCAD 2008-based products provide two options for lighting: standard (generic) lighting and photometric lighting. Previous versions offered only standard (generic) lighting. There is no explicit conversion required for lights from AutoCAD 2007 to AutoCAD 2008. Drawings from AutoCAD 2007 open by default in AutoCAD 2008 in the standard (generic) lighting workflow. The additional photometric properties available in AutoCAD 2008 are available as soon as the photometric lighting workflow is enabled in the drawing.

There is a conversion process required for drawings with lighting prior to AutoCAD 2007. AutoCAD-based products include a system variable, 3DCONVERSIONMODE, to automatically convert drawings with lighting from previous versions to the AutoCAD 2007 and AutoCAD 2008 lighting format.

Other lighting fidelity issues include

- **Lights in blocks.** Lights in blocks created in AutoCAD 2008 do not always display in AutoCAD 2007.
- **Texture illumination.** In AutoCAD 2007 and earlier releases, you cannot add lighting to textures as in AutoCAD 2008.
- **Ground Shadows.** The intensity of ground shadows in AutoCAD 2008 depends on the brightness of the light and angle of incidence.

AutoCAD Architecture (ACA) Custom Objects

In DWG TrueView, ACA (formerly called ADT) custom objects are **not** upgraded when converting the drawing containing them to a different drawing format. This is highly desirable.

In contrast, when the ACA Object Enabler is present in AutoCAD-based products, all ACA objects are upgraded to the latest ACA format. This can present problems when AutoCAD users on the latest version collaborate with ACA users that are several releases behind. AutoCAD users can open the ACA file and save it to the AutoCAD 2004 format used by the previous ACA version, but now the ACA objects are changed to the AutoCAD 2008 versions. When these upgraded ACA objects are encountered in ACA, they are treated as proxy objects.

Additional limitations of Saving to Earlier Versions

Converting a drawing to AutoCAD 2007 format is subject to the following limitations:

- DGN underlays do not display in releases prior to AutoCAD 2008.
- Multileaders display as proxy objects in releases prior to AutoCAD 2008.

Converting a drawing to AutoCAD 2004 format is subject to the following limitations:

- Dynamic block geometry can be redefined independent of the block's dynamic elements, and the geometry in the block reference is not updated when the drawing is opened in a product based on AutoCAD 2007 or later.

Converting a drawing to AutoCAD 2000 format is subject to the following limitations:

- File size can increase.
- Encryption and digital signatures are not preserved.

Converting a drawing in AutoCAD Release 14/AutoCAD LT 98/AutoCAD LT 97 format is subject to the following limitations:

- Hyperlinks are converted to AutoCAD Release 14 attached URLs.
- Database links and freestanding labels are converted to AutoCAD Release 14 links and displayable attributes.
- Database attached labels are converted to multiline text and leader objects, and their link information is not available. Attached labels are restored if you open the drawing in AutoCAD 2000 or later.
- Arc length and jogged dimension objects might not retain their original color in AutoCAD Release 14.
- DWG TrueView does not convert of pen-width settings to lineweights.

Quick Reference

DWGCONVERT

Converts drawing format version for selected drawing files.

Save Conversion Settings for Later Use

After you choose the conversion options you want, you can save time by saving the settings that you commonly use.

You will probably convert drawing files multiple times during a project. The DWG Convert feature provides a method to name and save your conversion settings. The Conversion Setups dialog box displays a list of saved conversion setups from which you can select each time you convert a set of files.

The default conversion setup is named Standard. Starting from this conversion setup, you can create one or more new conversion setups with appropriate names. Then you can modify or delete them as needed.

If you convert drawings using the same settings each time, then you can simply modify the Standard conversion setup to your requirements, and you do not need to save additional ones.

To specify an existing conversion setup

1. Click Application button ➤ Save As ➤ DWG Convert.
2. In the DWG Convert dialog box, under Select a Conversion Setup, click the conversion setup that you want to use.
3. (Optional) To examine the settings in a conversion setup, click Conversion Setups and, in the Conversion Setups dialog box, click Modify. Click Cancel when you are done.
4. Click Convert or Close.

To specify conversion setup options

1. Click Application button ➤ Save As ➤ DWG Convert.
2. In the DWG Convert dialog box, click Conversion Setups.
3. In the Conversion Setups dialog box, do *one* of the following:
   - Click a previously saved conversion setup from the list and click Close.
   - Click New to open the New Conversion Setup dialog box. After you enter a new name, click Continue to return to the Modify Conversion Setup dialog box.
   - Click a previously saved conversion setup and then click Modify. The Modify Conversion Setup dialog box is displayed.
4. (Optional) In the Modify Conversion Setup dialog box, specify all options that you want to change and click OK.
5. Click Close.

Quick Reference

DWGCONVERT

Converts drawing format version for selected drawing files.

Save Lists of Files for Batch Conversions

For repetitive conversions, the drawing files that you select for conversion can be saved to a list, called a *batch control list*.

Batch control lists are saved as BCL files that can also be opened and viewed in a text editor such as Notepad. Once you add several drawing files to the current list, you can remove any of them from the list by clearing its check box.

From the DWG Convert dialog box, you can also do the following:

- Add a drawing file to the current list, which is empty by default.
- Clear the list to start a new list.
- Open a previously saved batch control list.
- Append a previously saved batch control list to the current list.
- Save the current batch control list to the same or a new BCL file.

Using these controls, you can easily create and modify a set of batch control files.

To save a list of drawing files for repeated conversion

1. Click Application button ➤ Save As ➤ DWG Convert.
2. In the DWG Convert dialog box, near the bottom, click the Add File button.
3. In the Select File dialog box, navigate to the folder of the drawing file that you want to convert.
4. Click the drawing file name and click Open.
5. Repeat steps 2-4 for each drawing file to be added.
6. In the DWG Convert dialog box, click Save List.
7. In the Save Conversion List dialog box, enter a file name for the batch conversion list (BCL) file. Click Save.
8. In the DWG Convert dialog box, click Close.

Quick Reference

DWGCONVERT

Converts drawing format version for selected drawing files.

Import Other File Formats

You can import files, other than DWG files, that were created with other applications into your drawings.

Import Autodesk 3ds MAX Files

You can import a 3DS file created with Autodesk® 3ds MAX®.

Geometry and rendering data from 3ds MAX are imported, which includes meshes, materials, mappings, lights, and cameras. However, procedural materials, smoothing groups, and keyframe data cannot be imported. Imported objects are assigned a color as close as possible to the object's 3ds MAX color.

For lights and materials, use CONVERTOLDLIGHTS and CONVERTOLDMATERIALS. Lights are converted to the nearest equivalent.

- Ambient light loses its color.
- Omni lights become point lights.
- Spotlights become AutoCAD spotlights.
- Cameras become named views.

If the name of any 3ds MAX object conflicts with a name already in the AutoCAD drawing, the 3ds MAX name is assigned a sequence number to resolve the conflict. The name might be truncated to resolve the conflict.

To import an Autodesk 3ds MAX file

1. Click Insert ➤ 3D Studio. 

2. In the 3D Studio File Import dialog box, select the file that you want to import.
3. Click Open.

Quick Reference

Commands

3DSIN

Imports a 3D Studio (3DS) file.

IMPORT

Imports files of different formats into the current drawing.

Import FBX Files

FBX files allow you to import and export 3D objects, 2D objects with thickness, lights, cameras, and materials from one Autodesk program to another.

The FBX file format is an open framework for 3D data transfer that creates a high level of interoperability among Autodesk programs. For example, in Autodesk 3ds Max, you can export a file as an FBX file, and then open that file in AutoCAD to view and edit the objects, lights, cameras, and materials. Likewise, in AutoCAD, you can export a drawing as an FBX file, and then view and edit that file in 3ds Max.

Points (not point clouds), meshes, NURBS curves and surfaces, subdivided meshes, target cameras, lights, and materials are imported into AutoCAD in the same form in which they existed when exported from 3ds Max.

When importing FBX files from 3ds Max into AutoCAD,

- Target cameras created in 3ds Max are imported accurately.
- The LIGHTINGUNITS system variable controls whether lights are imported as standard or photometric lights.

When exporting FBX files from AutoCAD to 3ds Max,

- 3D objects, 2D objects with thickness, lights, cameras, and materials are rotated by 90 degrees on the Z axis.
- You can export 2D objects by increasing their thickness. Any visible and renderable object with thickness can be exported to FBX.

To export a 2D object to an FBX file, increase its thickness to convert to a 3D object. Any visible and renderable object or geometry with thickness can be exported to FBX.

To import an FBX file

1. In AutoCAD, click Insert tab ➤ Import panel ➤ Import. 
2. In the Import File dialog box, select the FBX file type, select the FBX file, and click Open.

   The FBX Import Options dialog box displays.
3. Specify the items to be imported, the assigned layer for the objects, and the conversion units. Importing the FBX file as a block allows you to modify it.
4. Click Ok to import the as specified.

Quick Reference

Command

FBXIMPORT

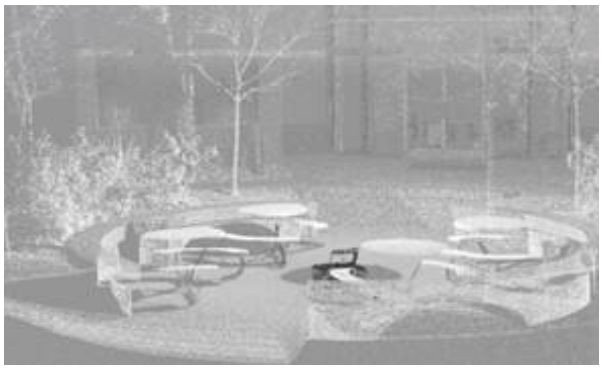
Imports an Autodesk® FBX file, which can contain objects, lights, cameras and materials.

Import 3D Point Cloud Files

Index and attach point cloud data created with 3D tripod and LiDAR scanners.

Index Point Cloud Data

A point cloud is a large collection of points placed on a 3D coordinate system. The points create a 3D representation of an object or geographic feature.



Create Point Cloud Files

AutoCAD can create 3D point clouds from the following point cloud data file types:

- FLS
- FWS
- LAS
- XYB

Index a file of one of these types to create either an ISD or PCG file in AutoCAD.

Work While Indexing a Point Cloud Data File

You can continue to use AutoCAD while your point cloud data file is being indexed. To view the status of indexing, hover the pointing device over the icon in the drawing status bar.



To cancel indexing, right-click the icon and click Cancel Indexing.



NOTE To attach a point cloud data file to an AutoCAD drawing, the data file must be indexed in AutoCAD.

Quick Reference

POINTCLOUD

Provides options to create and attach an indexed point cloud file.

POINTCLOUDINDEX

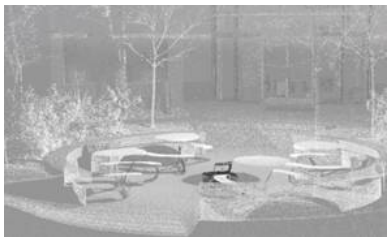
Creates an indexed point cloud (PCG or ISD) file from a scan file.

Attach a Point Cloud to a Drawing

When you attach a point cloud file to a drawing, you can specify whether it is locked for editing.

Insert Point Clouds with Geographic Data

If geographic location information exists in both the current drawing and the point cloud file, then you can insert the file based on that data.



Lock a Point Cloud from Editing

An indexed point cloud contains a Lock property that controls whether the point cloud can be selected and edited.

Quick Reference

POINTCLOUD

Provides options to create and attach an indexed point cloud file.

POINTCLOUDATTACH

Inserts an indexed point cloud (PCG or ISD) file into the current drawing.

POINTCLOUDAUTOUPDATE

Controls whether a point cloud is regenerated automatically after manipulation, panning, zooming, or orbiting.

POINTCLOUDDENSITY

Controls the number of points displayed at once for all point clouds in the drawing view.

POINTCLOUDLOCK

Controls whether an attached point cloud can be manipulated, moved, or rotated.

POINTCLOUDRTDENSITY

Improves performance by degrading the number of points displayed in the drawing view zooming, panning, or orbiting in real time.

Work with 3D Point Clouds

You can use several common commands to edit, or add geometry to, a 3D point cloud.

When an attached point cloud file is unlocked, you can click the point cloud to display an editing grip at the insertion point of the point cloud.

The default insertion point is the origin of the coordinate system (0, 0, 0) of the indexed file. If, in the indexed file, a point cloud is offset from the origin, when attached, the insertion point is also offset from the points of the point cloud.

To align point clouds, enter the same insertion point, rotation, and scale for each point cloud.



Edit a Point Cloud

Use the following commands to modify a point cloud:

- BLOCK
- COPY
- COPYCLIP
- CUTCLIP
- ERASE
- MOVE
- PASTECLIP
- ROTATE
- SCALE
- STRETCH

NOTE The MIRROR3D command copies and moves a point cloud, but does not mirror it. You cannot explode a point cloud.

Snap to Points on a Point Cloud

The Node object snap allows you to snap to individual points on a point cloud. The Insertion object snap allows you to snap to the insertion point of the point cloud. For more information, see Object Snap Modes.

Quick Reference

POINTCLOUD

Provides options to create and attach an indexed point cloud file.

POINTCLOUDAUTOUPDATE

Controls whether a point cloud is regenerated automatically after manipulation, panning, zooming, or orbiting.

POINTCLOUDDENSITY

Controls the number of points displayed at once for all point clouds in the drawing view.

POINTCLOUDLOCK

Controls whether an attached point cloud can be manipulated, moved, or rotated.

POINTCLOUDRTDENSITY

Improves performance by degrading the number of points displayed in the drawing view zooming, panning, or orbiting in real time.

Import ACIS SAT Files

You can import geometry objects stored in SA (ASCII) files using the ASCISIN command.

ACISIN converts the model to a body object or to 3D solids and regions if the body is a true solid or a true region.

Quick Reference

Commands

ACISIN

Imports an ACIS (SAT) file and creates 3D solid, body, or region objects.

IMPORT

Imports files of different formats into the current drawing.

Convert DXF and DXB Files to DWG Format

DXF and DXB files are two types of drawing interchange files used to transfer data between various applications.

A DXF (drawing interchange format) file is either a binary or an ASCII representation of a drawing file. It is often used to share drawing data between other CAD programs.

A DXB (drawing interchange binary) file is a specially coded binary version of a DXF file used for plotting, and can be used to "flatten" 3D wireframe drawings into 2D vectors.

You can convert a DXF or DXB file to DWG format by opening the file and saving it in DWG format. You can then work with the resulting drawing file as you would with any other drawing file.

To open a DXF file

1. Click File menu ➤ Open.
2. In the Select File dialog box, in the Files of type box, select DXF (*.dxf).
3. Find and select the DXF file you want to import, or enter the name of the DXF file at File Name.
4. Click Open.

To open a DXB file

1. Click Insert ➤ Drawing Exchange Binary. 

2. In the Select DXB File dialog box, find and select the DXB file you want to import.
3. Click Open.

Quick Reference

Commands

DXBIN

Imports an AutoCAD DXB (drawing interchange binary) file.

OPEN

Opens an existing drawing file.

Import MicroStation DGN Files

You can import MicroStation® DGN drawing files into DWG files.

The import process translates basic DGN data into the corresponding DWG file data. There are several translation options to determine how certain data such as text elements and external references are handled. Users can not only manipulate text elements and xrefs, but also select the conversion units based on DGN unit settings, and most importantly, preview the default mapping translation or create and use their own mapping setups (see DGNMAPPING).

Exchanging and reusing basic drawing data is useful in collaborative projects. For example, service organizations such as AEC and design-build firms might

need to import mapping data created with MicroStation into a site plan created with an AutoCAD-based product. The DGN data can serve as an accurate reference for creating the site plan.

NOTE DGNIMPORT is not limited to files with *.dgn extensions. It supports all DGN files, even those that do not have a .dgn extension.

Understand the Limitations

The DGN import and export capabilities are designed to provide a fundamental exchange of information between MicroStation V7/V8 DGN files and AutoCAD DWG files. However, translating data from one format to a completely different format inevitably require compromises and substitutions.

- Simple geometric objects such as lines, arcs, and circles, and properties such as layer assignments correlate directly between the DWG and DGN data formats.
- Data with built-in features or variations are visually approximated. For example, text and dimensions might have specialized formatting, and color definitions might be customized.
- Some data cannot be translated completely. For example, product-specific features such as data fields or dynamic blocks can be represented visually but not behaviorally.
- AutoCAD only supports colors for lines so all entities in the DGN file that have a color property set for a symbol in MicroStation lose the symbol color when imported as a block.
- When importing DGN files that contain multilines, any linetypes associated with the multilines do not display.
- Ensure that you create uniformly scaled blocks in MicroStation. Entities with DGN line styles in non-uniformly scaled blocks display differently in MicroStation.
- All line style modifier settings of an element in the DGN file are lost when exporting them from AutoCAD back to MicroStation.
- The closed property of a DGN polygon or closed polyline is automatically set to No when importing a DGN file into AutoCAD. This is done to ensure that linetypes display properly.

See the DGNIMPORT Conversion Table for specific details about the scope of importing MicroStation objects and other data.

See the DGNIMPORT Unit Mapping Table for specific details about unit conversions when importing MicroStation objects.

Recommendations

To optimize the data transfer from the MicroStation® DGN file format, request that the creator of the MicroStation drawings consider the following recommendations:

- Create a separate DGN file for each design model
- Do not reference sheet models from design models
- Minimize the use of custom objects and other data unique to MicroStation
- Use ByLevel for color, line style, and weight properties
- Use simple patterns and pattern styles
- Use TrueType text fonts rather than SHX text fonts
- Use defined styles for dimensions and text rather than a "none" style
- Use simple dimension styles if possible
- Use shared cells as opposed to normal (library) cells
- Work primarily in one format, DWG or DGN, rather than switching back and forth

When importing a DGN file, specify the conversion units (master units or sub-units) based on the DWG file's drawing units. For example, if you receive a DGN file with master units set to meters, and sub-units set to millimeters, and you need the drawing units of the DWG file to be in meters, then select master units in the Import DGN Settings dialog box. This matches the meters in the DGN file to the meters in the DWG file.

Import Custom Line Style Definitions

If a DGN file does not contain linetype definitions and these definitions do not exist in any .rsc files in the support path, then all linetypes in the DGN file are displayed as a continuous linetype in AutoCAD. To ensure that custom linetypes are displayed, copy the corresponding .rsc files to the support path or add the RSC file path to the search path in AutoCAD in the Options dialog box (Files tab, Support File Search Path).

The RSC files are specified by the MS_SYMBRSRC configuration variable setting in the directory specified in the MS_RSRCPATH setting in MicroStation.

Import DGN Settings

In the Import DGN Settings dialog box, if you select "Import into Current Drawing" and select the "Translate References to Blocks or Xrefs" option in the External DGN references section, all referenced DGN files are converted to block references. If you do not select "Import into Current Drawing" and select the "Translate References to Blocks or Xrefs" option, then all referenced DGN files are converted to DWG external references.

Microstation Standard Line Styles

You can attach a DGN file as an underlay or import the DGN fiel into a drawing. It is recommended that you do not use the MicroStation standard line styles 1 to 7 because the scaling for these line styles is based on output device coordinates. For best results, use custom line styles.

Manage Translation Mapping Setups

You can manage translation mapping setups for DGN file import and export operations. You can create, modify, rename, or delete mapping translations based on your company's CAD standards such as:

- Change DGN level names to appropriate DWG layer names
- Remap unsupported DGN linestyles to DWG linetypes
- Remap lineweights and adjust color mapping

Therefore, you can streamline the import/export process while minimizing the need for more extensive editing.

The DGN Mapping Setups dialog box is displayed when you click the Mapping Setups button in the Import DGN Settings and Export DGN Settings dialog boxes. You can also enter DGNMAPPING at the Command prompt.

To import a MicroStation DGN drawing

1. Click Insert tab ➤ Import panel ➤ Import. 
2. In the Import File dialog box, in the Files of type box, select MicroStation DGN (*.dgn).
3. Find and select the DGN file you want to import, or enter the name of the DGN file at File Name.
4. Click Open.

5 In the Import DGN Settings dialog box, select a design model from the list and specify the desired import options.

6 Click OK.

To create a new DGN mapping setup

1 In Import (or Export) DGN Settings dialog box, click Mapping Setups. The DGN Mapping Setups dialog box is displayed.

2 Click New. The New Mapping Setup dialog box is displayed.

3 Under New DGN mapping setup name, enter a mapping setup name.

4 Under Based On, select a mapping setup. If there is no existing mapping setup, the default is Standard.

5 Select a mapping type.

6 Click Continue. The Modify DGN Mapping Setup dialog box is displayed.

7 Specify property mapping for DGN Import (or Export).

8 Click Ok.

To rename a DGN mapping setup

1 In the DGN Mapping Setups dialog box , select the mapping setup you want to rename.

NOTE The Rename button is disabled when the Standard mapping setup is selected.

2 Click Rename.

3 Enter new mapping setup name.

4 Click anywhere outside the text box.

You can also right-click the mapping setup name from the mapping setup list, click Rename and then continue with step 3.

To modify a DGN mapping setup

1 In the DGN Mapping Setups dialog box , select the mapping setup you want to modify.

NOTE The Modify button is disabled when the Standard mapping setup is selected.

2. Click Modify. The Modify DGN Mapping Setup: <mapping_setup> dialog box is displayed.
3. Specify property mapping for DGN Import (or Export).
4. Click Ok.

You can also right-click the mapping setup name from the mapping setup list, click Modify and then continue with step 3.

To delete a DGN mapping setup

1. In the DGN Mapping Setups dialog box , select the mapping setup you want to delete.

NOTE The Delete button is disabled when the Standard mapping setup is selected.

2. Click Delete.
3. When the Delete Mapping Setup prompt is displayed, click Yes.

You can also right-click the mapping setup name from the mapping setup list and click Delete.

To add properties from a DGN file

1. In the DGN Mapping Setups dialog box , click Modify.

NOTE The Modify button is disabled when the Standard mapping setup is selected.

2. In the Modify DGN Mapping Setup dialog box, click Add Properties from DGN File button.
3. The Add Properties from DGN File dialog box is displayed. Select a DGN file and click Open.

The unique properties of the file are added to the bottom of the DGN column for each tab (Level, Linestyle, Lineweight, and Color).

NOTE Duplicate properties will be ignored.

To add properties from a drawing file

1. In the DGN Mapping Setups dialog box , click Modify.

   NOTE The Modify button is disabled when the Standard mapping setup is selected.

2. In the Modify DGN Mapping Setup dialog box, click Add Properties from Drawing File button.
3. The Add Properties from Drawing File dialog box is displayed. Select a drawing file and click Open.

The unique properties of the file are added to the bottom of the DWG column for each tab (Level, Linestyle, Lineweight, and Color).

NOTE Duplicate properties will be ignored.

Quick Reference

Commands

DGNIMPORT

Imports the data from a DGN file into a new DWG file or the current DWG file, depending on the DGNIMPORTMODE system variable.

DGNMAPPING

Specifies the location of the dgnsetups.ini file where DGN mapping setups are stored.

System Variables

DGNIMPORTMAX

Sets the maximum number of elements that are translated when importing a DGN file.

DGNIMPORTMODE

Controls the default behavior of the DGNIMPORT command.

DGNMAPPINGPATH

Specifies the location of the dgnsetups.ini file where DGN mapping setups are stored.

Import IGES Files

You can import an IGES (*.igs or *.iges) file into a DWG file.

Initial Graphics Exchange Specification (IGES) is a neutral file format designed to transfer 2D and 3D drawing data between dissimilar CAD systems. The IGES standard defines two file formats: fixed-length ASCII, which stores information in 80-character records, and compressed ASCII. The IGESIMPORT and IMPORT commands support only the fixed-length format.

Depending on how large and complex the IGES file is, IGESIMPORT may take time to process the file. Hence, IGESIMPORT starts running in the background and displays an import icon on the status bar. Consequently, you can continue to work on the current drawing while IGESIMPORT processes the IGES file. The tooltip that is displayed when you place the cursor over the Import status bar icon indicates the current state of the import process.

Once processing is complete, a notification bubble is displayed on the import status bar icon. If you click the link in the notification bubble, the imported content is inserted in the current drawing. If you close the notification bubble before you insert the imported content, you can right-click the import status bar icon and choose “Insert” from the menu.

Understanding the Limitations

Under ideal conditions, translation preserves the appearance and functionality of entities. However, this process has limitations, and some data will not be preserved in a round trip between different CAD systems.

For example, when exporting to IGES, a 2D polyline is translated as IGES entity 106:12. When importing from IGES, entity 106:12 translates to a spline. Hence, if you use IGESIMPORT to read a file created by IGESEXPORT, the resulting drawing may not be identical to the original drawing.

Similar data loss may occur when transferring data between AutoCAD and another CAD system, using IGES. Usually, a one-time translation to or from IGES does not pose a serious problem. However, data loss can become a concern if drawings are repeatedly translated in and out of IGES.

Recommendations

Inspect the conversion tables of the IGES translators of both the source CAD program and AutoCAD. Get to know what entities are supported. Restructure the drawing to minimize the usage of objects that require compromises and substitutions.

See the conversion table for IGESIMPORT for specific details of the scope of the IGES translator.

To import an IGES file

1. Click Insert tab ➤ Import panel ➤ Import. 
2. In the Import File dialog box, in the Files of type box, select (*.igs, *.iges).
3. Find and select the IGES file you want to import, or enter the name of the IGES file at File Name.
4. Click Open.

   If translation takes longer than 5 seconds, import is performed as a background process. A notification bubble is displayed when the command is done.

   - Click the file name in the notification bubble. The imported drawing data is inserted in the current drawing.

   TIP If you accidentally close the notification bubble, right-click the import icon on the status bar, and click Insert. The imported IGES file is inserted in the current drawing.

5. Click OK.

Quick Reference

Commands

IGESIMPORT

Imports data from an IGES (*.igs or *.iges) file into the current drawing.

IMPORT

Imports files of different formats into the current drawing.

Import STEP Files

You can import STEP (*.stp, *.ste, or *.step) files and insert them in the current drawing.

Standard for the Exchange of Product model data (STEP) is an ISO standard for computer interpretable product manufacturing data representation (CAD data as well as meta data) and exchange. STEP is often used for exchanging 3D model data between dissimilar CAD systems. The IMPORT command supports STEP versions AP214 and AP203E2.

The following data types are translated:

- Assemblies
- Parts
- Surfaces
- Solids
- Wire Geometry (2D & 3D)

All geometry is created on layer 0. The original colors for all 2D and 3D wire geometry are preserved. They do not acquire the color from layer 0 (color is not ByLayer).

IMPORT maps all STEP assembly structures to blocks and block references. Individual part in an input STEP file is mapped to a block, which in turn is referenced from model space. Assemblies are formed by nesting "part" blocks. The name of the block is the same as the name of the corresponding part or assembly in STEP. If a STEP part or assembly name contains unsupported characters, in the corresponding block they are replaced by the underscore (_) character. If a part or assembly name is longer than the maximum block name length, the name is truncated. If this results in identically named blocks, a unique name is assigned to one of the blocks by appending the dollar character ($) to its name, along with a unique number. The original name is preserved in the block description.

The INSUNITS property on the block corresponding to the top level assembly is set to match the source file's linear units. If the source file's units cannot be found or is not supported by INSUNITS, the property is set to "No Units".

Depending on how complex the STEP file is, IMPORT could take several minutes to execute. Hence, IMPORT runs as a background process, allowing you to continue working on the drawing. Once the background process starts, IMPORT displays the Import icon on the status bar. If you hover over the

Import status bar icon while import is in progress, a tooltip displays the current status.

Once processing is complete, a notification bubble is displayed on the import status bar icon. If you click the link in the notification bubble, the imported content is inserted in the current drawing. If you close the notification bubble before you insert the imported content, you can insert it later by right-clicking the import status bar icon and choosing "Insert" from the menu.

To import a STEP file

1. Click Insert tab ➤ Import panel ➤ Import. 
2. In the Import File dialog box, in the Files of type box, select (*.stp, *.step).
3. Find and select the STEP file you want to import, or enter the name of the STEP file at File Name.
4. Click Open.

   Import is performed as a background process. A notification bubble is displayed when the command is done.

   - Click the file name in the notification bubble. The imported STEP data is inserted in the current drawing.

   TIP If you accidentally close the notification bubble, right-click the import icon on the status bar, and click Insert. The imported STEP file is inserted in the current drawing.

5. Click OK.

Quick Reference

Commands

IMPORT

Imports files of different formats into the current drawing.

Import Pro/ENGINEER Files

You can import models created in Pro/ENGINEER® and insert them in the current drawing.

The following Pro/ENGINEER file types can be imported using the IMPORT command:

- *.prt* (part) (up to version 4.0)
- *.asm* (assembly) (up to version 4.0)
- *.g (Granite) (up to version 5.0)
- *.neu* (Neutral)

The following data types are translated:

- Surfaces
- Solids
- Multi Solids
- Wire Geometry (2D & 3D)

All geometry is created on layer 0. The original colors for all 2D and 3D wire geometry are preserved. They do not acquire the color from layer 0 (color is not ByLayer).

Importing *.prt*, *.g, *.neu* files produces blocks that have the same name as the source file. If the file name contains invalid characters, in the corresponding block they are replaced by the underscore (_) character. The INSUNITS property on the block is set to match the source file's linear units.

Importing *.asm* files produces nested blocks that have the same name as the source file. Each part is mapped to a block and "assembly" blocks are generated by nesting "part" blocks. The INSUNITS property on the block corresponding to the top level assembly is set to match the source file's linear units.

If the source file's units cannot be found or is not supported by INSUNITS, the property is set to "No Units".

Depending on how complex the Pro/ENGINEER file is, IMPORT could take several minutes to execute. Hence, IMPORT runs as a background process, allowing you to continue working on the drawing. Once the background process starts, IMPORT displays the Import icon on the status bar. If you hover

over the Import status bar icon while import is in progress, a tooltip displays the current status.

Once processing is complete, a notification bubble is displayed on the import status bar icon. If you click the link in the notification bubble, the imported content is inserted in the current drawing. If you close the notification bubble before you insert the imported content, you can insert it later by right-clicking the import status bar icon and choosing "Insert" from the menu.

To import a Pro/ENGINEER file

1 Click Insert tab ➤ Import panel ➤ Import. 

2 In the Import File dialog box, in the Files of type box, select the file type corresponding to the Pro/ENGINEER file you want to import (*.prt*, *.asm*, *.g, *.neu*).

NOTE SolidWorks, NX, and Pro/ENGINEER files share the same file extension (*.prt, *.asm). Hence make sure that you select the correct file type.

3 Find and select the file you want to import, or enter the name of the file at File Name.

4 Click Open.

Import is performed as a background process. A notification bubble is displayed when the command is done.

- Click the file name in the notification bubble. The imported file is inserted in the current drawing.

TIP If you accidentally close the notification bubble, right-click the import icon on the status bar, and click Insert. The imported file is inserted in the current drawing.

5 Click OK.

Quick Reference

Commands

IMPORT

Imports files of different formats into the current drawing.

Import Rhino Files

You can import Rhinoceros®(Rhino) models and insert them in the current drawing.

Rhinoceros (Rhino) is a stand-alone, commercial non-uniform rational basis spline (NURBS) based 3-D modeling tool. The Rhino file format (*.3dm) is often used for the exchange of NURBS geometry between 3D CAD systems. You can use the IMPORT command to import a *.3dm file and insert the surfaces into the current drawing.

All geometry is created on layer 0. The original colors for all 2D and 3D wire geometry are preserved. They do not acquire the color from layer 0 (color is not ByLayer).

Importing a *.3ds file produces a block that has the same name as the source file. If the file name contains invalid characters, in the corresponding block they are replaced by the underscore (_) character. The INSUNITS property on the block is set to match the source file's linear units. If the source file's units cannot be found or is not supported by INSUNITS, the property is set to "No Units".

Depending on how complex the Rhino file is, IMPORT could take several minutes to execute. Hence, IMPORT runs as a background process, allowing you to continue working on the drawing. Once the background process starts, IMPORT displays the Import icon on the status bar. If you hover over the Import status bar icon while import is in progress, a tooltip displays the current status.

Once processing is complete, a notification bubble is displayed on the import status bar icon. If you click the link in the notification bubble, the imported content is inserted in the current drawing. If you close the notification bubble before you insert the imported content, you can insert it later by right-clicking the import status bar icon and choosing "Insert" from the menu.

To import a Rhino file

1. Click Insert tab ➤ Import panel ➤ Import. 
2. In the Import File dialog box, in the Files of type box, select the file type corresponding to the Rhino file you want to import (*.3dm).
3. Find and select the file you want to import, or enter the name of the file at File Name.
4. Click Open.

Import is performed as a background process. A notification bubble is displayed when the command is done.

- Click the file name in the notification bubble. The imported file is inserted in the current drawing.

TIP If you accidentally close the notification bubble, right-click the import icon on the status bar, and click Insert. The imported file is inserted in the current drawing.

5 Click OK.

Quick Reference

Commands

IMPORT

Imports files of different formats into the current drawing.

Import CATIA Files

You can import CATIA® V4 and CATIA® V5 models and insert them in the current drawing.

CATIA (Computer Aided Three-dimensional Interactive Application) is a multi-platform CAD/CAM/CAE commercial software suite. CATIA V4 runs on UNIX variants, while CATIA V5 runs on Windows as well. The file formats used by CATIA V4 and CATIA V5 are different. The IMPORT command supports import from both CATIA V4 (all releases) and CATIA V5 (release 9 - 20) files and inserts the model into the current drawing.

CATIA V4

The following CATIA V4 files are supported

- *.model (model files)
- *.session (session files)
- *.exp and *.dlv (export files)

IMPORT maps all CATIA V4 assembly structures to blocks and block references. Each individual part in an input CATIA V4 file is mapped to a block, which

in turn is referenced from model space. Assemblies are formed by nesting "part" blocks.

CATIA V4 models can contain components that are set to be invisible/hidden. However, once imported, all geometry is visible, regardless of their visibility state in CATIA V4.

A CATIA V4 *.exp file can contain more than one model. However, you can import only one model per IMPORT session. Hence, when IMPORT detects multiple models, it prompts you to specify which model to import.

CATIA V5

The following CATIA V5 files are supported

- *.CATPart (CATIA part files)
- *.CATProduct (CATIA assembly files)

Importing *.CATPart files produces blocks that have the same name as the source file. If the file name contains invalid characters, in the corresponding block they are replaced by the underscore (_) character.

The Import Process

Depending on how complex the CATIA file is, IMPORT could take several minutes to execute. Hence, IMPORT runs as a background process, allowing you to continue working on the drawing. Once the background process starts, IMPORT displays the Import icon on the status bar. If you hover over the Import status bar icon while import is in progress, a tooltip displays the current status.

The following data types are translated:

- Assemblies
- Parts
- Surfaces
- Solids
- Wire Geometry (2D & 3D)

Once processing is complete, a notification bubble is displayed on the import status bar icon. If you click the link in the notification bubble, the imported content is inserted in the current drawing. If you close the notification bubble

before you insert the imported content, you can insert it later by right-clicking the import status bar icon and choosing "Insert" from the menu.

When IMPORT produces a single block, the INSUNITS property on the block is set to match the source file's linear units. When IMPORT produces multiple blocks, the INSUNITS property on the block corresponding to the top level assembly is set to match the source file's linear units.

To import a CATIA V4 file

1 Click Insert tab ➤ Import panel ➤ Import. 

2 In the Import File dialog box, in the Files of type box, select the file type corresponding to the CATIA V4 file you want to import (*.model, *.session, *.exp, *.dlv3).

3 Find and select the file you want to import, or enter the name of the file at File Name.

4 Click Open.

Import is performed as a background process. A notification bubble is displayed when the command is done.

- Click the file name in the notification bubble. The imported file is inserted in the current drawing.

TIP If you accidentally close the notification bubble, right-click the import icon on the status bar, and click Insert. The imported file is inserted in the current drawing.

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5 Click OK.

To import a CATIA V5 file

1 Click Insert tab ➤ Import panel ➤ Import. 

2 In the Import File dialog box, in the Files of type box, select the file type corresponding to the CATIA V5 file you want to import (*.CATPart, *.CATProduct).

3 Find and select the file you want to import, or enter the name of the file at File Name.

4 Click Open.

Import is performed as a background process. A notification bubble is displayed when the command is done.

- Click the file name in the notification bubble. The imported file is inserted in the current drawing.

TIP If you accidentally close the notification bubble, right-click the import icon on the status bar, and click Insert. The imported file is inserted in the current drawing.

5 Click OK.

Quick Reference

Commands

IMPORT

Imports files of different formats into the current drawing.

Import SolidWorks Files

You can import models created in SolidWorks® and insert them in the current drawing.

The following SolidWorks file types can be imported using the IMPORT command:

- *.prt, *.sldprt (part) (versions 2003 - 2010)
- *.asm, *.sldasm (assembly) (versions 2003 -2010)

The following data types are translated:

- Assemblies
- Parts
- Surfaces
- Solids
- Multi Solids

- Wire Geometry (2D & 3D)

All geometry is created on layer 0. The original colors for all 2D and 3D wire geometry are preserved. They do not acquire the color from layer 0 (color is not ByLayer).

Importing part files produces blocks that have the same name as the source file. If the file name contains invalid characters, in the corresponding block they are replaced by the underscore (_) character. The INSUNITS property on the block is set to match the source file's linear units.

Importing assembly files produces nested blocks that have the same name as the source file. Each part is mapped to a block and "assembly" blocks are generated by nesting "part" blocks. The INSUNITS property on the block corresponding to the top level assembly is set to match the source file's linear units.

If the source file's units cannot be found or is not supported by INSUNITS, the property is set to "No Units".

Depending on how complex the SolidWorks file is, IMPORT could take several minutes to execute. Hence, IMPORT runs as a background process, allowing you to continue working on the drawing. Once the background process starts, IMPORT displays the Import icon on the status bar. If you hover over the Import status bar icon while import is in progress, a tooltip displays the current status.

Once processing is complete, a notification bubble is displayed on the import status bar icon. If you click the link in the notification bubble, the imported content is inserted in the current drawing. If you close the notification bubble before you insert the imported content, you can insert it later by right-clicking the import status bar icon and choosing "Insert" from the menu.

To import a SolidWorks file

1. Click Insert tab ➤ Import panel ➤ Import. 
2. In the Import File dialog box, in the Files of type box, select the file type corresponding to the SolidWorks file you want to import (*.prt, *.sldprt, *.asm, *.sldasm).

NOTE SolidWorks, NX, and Pro/ENGINEER files share the same file extension (*.prt, *.asm). Hence make sure that you select the correct file type.

3 Find and select the file you want to import, or enter the name of the file at File Name.

4 Click Open.

Import is performed as a background process. A notification bubble is displayed when the command is done.

- Click the file name in the notification bubble. The imported file is inserted in the current drawing.

TIP If you accidentally close the notification bubble, right-click the import icon on the status bar, and click Insert. The imported file is inserted in the current drawing.

5 Click OK.

Quick Reference

Commands

IMPORT

Imports files of different formats into the current drawing.

Import JT Files

You can import JT models and insert them in the current drawing.

JT is a 3D data format used for product visualization, collaboration, and CAD data exchange. The IMPORT command supports JT versions versions 7.0, 8.0, 8.1, 8.2, and 9.0 - 9.5.

The following data types are translated:

- Assemblies
- Parts
- Surfaces
- Solids
- Multi Solids
- Wire Geometry (2D & 3D)

All geometry is created on layer 0. The original colors for all 2D and 3D wire geometry are preserved. They do not acquire the color from layer 0 (color is not ByLayer).

All geometry is created on layer 0. The original colors for all 2D and 3D wire geometry are preserved. They do not acquire the color from layer 0 (color is not ByLayer).

IMPORT maps all assembly structures to blocks and block references. Each individual part in an input JT file is mapped to a block, which in turn is referenced from model space. Assemblies are formed by nesting "part" blocks. The name of the block is the same as the name of the corresponding part or assembly in the JT file. If a part or assembly name contains unsupported characters, in the corresponding block they are replaced by the underscore (_) character. If a part or assembly name is longer than the maximum block name length, the name is truncated. If this results in identically named blocks, a unique name is assigned to one of the blocks by appending the dollar character ($) to its name, along with a unique number. The original name is preserved in the block description.

The INSUNITS property on the block corresponding to the top level assembly is set to match the source file's linear units. If the source file's units cannot be found or is not supported by INSUNITS, the property is set to "No Units".

Depending on how complex the JT model is, IMPORT could take several minutes to execute. Hence, IMPORT runs as a background process, allowing you to continue working on the drawing. Once the background process starts, IMPORT displays the Import icon on the status bar. If you hover over the Import status bar icon while import is in progress, a tooltip displays the current status.

Once processing is complete, a notification bubble is displayed on the import status bar icon. If you click the link in the notification bubble, the imported content is inserted in the current drawing. If you close the notification bubble before you insert the imported content, you can insert it later by right-clicking the import status bar icon and choosing "Insert" from the menu.

To import a JT file

1. Click Insert tab ➤ Import panel ➤ Import. 
2. In the Import File dialog box, in the Files of type box, select JT (*.jt).
3. Find and select the file you want to import, or enter the name of the file at File Name.

4 Click Open.

Import is performed as a background process. A notification bubble is displayed when the command is done.

- Click the file name in the notification bubble. The imported file is inserted in the current drawing.

TIP If you accidentally close the notification bubble, right-click the import icon on the status bar, and click Insert. The imported file is inserted in the current drawing.

5 Click OK.

Quick Reference

Commands

IMPORT

Imports files of different formats into the current drawing.

Import UGS NX Files

You can import NX models and insert them in the current drawing.

The following NX file types can be imported using the IMPORT command:

- *.prt (part)
- *.prt (assembly)

The IMPORT command supports NX versions 3 - 7.5. The following data types are translated:

- Assemblies
- Parts
- Surfaces
- Solids
- Multi Solids
- Wire Geometry (2D & 3D)

All geometry is created on layer 0. The original colors for all 2D and 3D wire geometry are preserved. They do not acquire the color from layer 0 (color is not ByLayer).

All geometry is created on layer 0. The original colors for all 2D and 3D wire geometry are preserved. They do not acquire the color from layer 0 (color is not ByLayer).

IMPORT maps all assembly structures to blocks and block references. Each individual part in an input NX file is mapped to a block, which in turn is referenced from model space. Assemblies are formed by nesting "part" blocks. The name of the block is the same as the name of the corresponding part or assembly in the NX file. If a part or assembly name contains unsupported characters, in the corresponding block they are replaced by the underscore (_) character. If a part or assembly name is longer than the maximum block name length, the name is truncated. If this results in identically named blocks, a unique name is assigned to one of the blocks by appending the dollar character ($) to its name, along with a unique number. The original name is preserved in the block description.

The INSUNITS property on the block corresponding to the top level assembly is set to match the source file's linear units. If the source file's units cannot be found or is not supported by INSUNITS, the property is set to "No Units".

Depending on how complex the NX model is, IMPORT could take several minutes to execute. Hence, IMPORT runs as a background process, allowing you to continue working on the drawing. Once the background process starts, IMPORT displays the Import icon on the status bar. If you hover over the Import status bar icon while import is in progress, a tooltip displays the current status.

Once processing is complete, a notification bubble is displayed on the import status bar icon. If you click the link in the notification bubble, the imported content is inserted in the current drawing. If you close the notification bubble before you insert the imported content, you can insert it later by right-clicking the import status bar icon and choosing "Insert" from the menu.

To import a NX file

1 Click Insert tab ➤ Import panel ➤ Import. 

2 In the Import File dialog box, in the Files of type box, select NX (*.prt).

NOTE SolidWorks, NX, and Pro/ENGINEER files share the same file extension (*.prt). Hence make sure that you select the correct file type.

3 Find and select the file you want to import, or enter the name of the file at File Name.

4 Click Open.

Import is performed as a background process. A notification bubble is displayed when the command is done.

- Click the file name in the notification bubble. The imported file is inserted in the current drawing.

TIP If you accidentally close the notification bubble, right-click the import icon on the status bar, and click Insert. The imported file is inserted in the current drawing.

5 Click OK.

Quick Reference

Commands

IMPORT

Imports files of different formats into the current drawing.

Import Parasolid Files

You can import Parasolid (*.x_b or *.x_t) files and insert them in the current drawing.

The following Parasolid file types can be imported using the IMPORT command:

- *.x_t (part) (up to version 23)
- *.x_b (binary) (up to version 23)

The following data types are translated:

- Surfaces
- Solids
- Wire Geometry (2D & 3D)

All geometry is created on layer 0. The original colors for all 2D and 3D wire geometry are preserved. They do not acquire the color from layer 0 (color is not ByLayer).

IMPORT maps all assembly structures to blocks and block references. Individual parts in an input Parasolid file is mapped to a block, which in turn is referenced from model space. Assemblies are formed by nesting "part" blocks. The name of the block is the same as the name of the corresponding Parasolid component. If the component name contains unsupported characters, in the corresponding block they are replaced by the underscore (_) character. If a component name is longer than the maximum block name length, the name is truncated. If this results in identically named blocks, a unique name is assigned to one of the blocks by appending the dollar character ($) to its name, along with a unique number. The original name is preserved in the block description.

When IMPORT produces a single block, the INSUNITS property on the block is set to match the source file's linear units. When IMPORT produces multiple blocks, the INSUNITS property on the block corresponding to the top level assembly is set to match the source file's linear units.

Depending on how complex the Parasolid file is, IMPORT could take several minutes to execute. Hence, IMPORT runs as a background process, allowing you to continue working on the drawing. Once the background process starts, IMPORT displays the Import icon on the status bar. If you hover over the Import status bar icon while import is in progress, a tooltip displays the current status.

Once processing is complete, a notification bubble is displayed on the import status bar icon. If you click the link in the notification bubble, the imported content is inserted in the current drawing. If you close the notification bubble before you insert the imported content, you can insert it later by right-clicking the import status bar icon and choosing "Insert" from the menu.

To import a Parasolid file

1. Click Insert tab ➤ Import panel ➤ Import. 
2. In the Import File dialog box, in the Files of type box, select (*.x_b, *.x_t).
3. Find and select the Parasolid file you want to import, or enter the name of the Parasolid file at File Name.
4. Click Open.

Import is performed as a background process. A notification bubble is displayed when the command is done.

- Click the file name in the notification bubble. The imported Parasolid data is inserted in the current drawing.

TIP If you accidentally close the notification bubble, right-click the import icon on the status bar, and click Insert. The imported Parasolid file is inserted in the current drawing.

5 Click OK.

Quick Reference

Commands

IMPORT

Imports files of different formats into the current drawing.

Insert WMF Files

WMF (Windows metafile format) files are frequently used to produce clip art and other nontechnical images that you need for drawings. You can insert a WMF file into a drawing file as a block. Unlike bitmaps, WMF files contain vector information that can be resized and printed without losing resolution. If the WMF file contains 2D solids or wide lines, you can turn off their display to increase drawing speed.

WMF files can contain both vector and raster information. However, the program uses only the vector information from a WMF file. When a WMF file containing raster information is imported into the program, the raster information is ignored.

See also:

- Import Text from External Files on page 1438

To insert a WMF file

1 Click Insert ➤ Windows Metafile. 

2 In the Import WMF dialog box, select the WMF file you want to open. Click Open.

3 Specify an insertion point for the WMF file.

4 To specify the scaling, use *one* of the following methods:

- To specify an imaginary box whose dimensions correspond to the scale factor, enter **c** (Corner) at the Command prompt. Specify the opposite corner above and to the right of the insertion point to avoid inserting a mirror image.
- To specify 3D scale factors, enter **xyz**.
- To set the *X* and *Y* scale settings, enter a scale at the Command prompt. You can set different horizontal and vertical scales. Enter **1** to retain the current scale. Enter a higher number to enlarge the metafile image. (Enter **2** to double the size of the image.) Enter a positive number less than 1 to shrink the image. (Enter **.5** to halve the size of the image.) Note that using this option makes it very difficult to import the WMF file at a specific scale or size.

5 Specify the rotation angle.

To control the display of solid fills and wide lines in WMF files

1 Click Blocks & References tab ➤ Import panel ➤ WMF File. 

2 In the Import WMF dialog box, click Tools ➤ Options.

3 In the WMF In Options dialog box, select or clear Wire Frame (No Fills) and Wide Lines to set the display of solid fills and wide lines. By default, these options are on.

4 Click OK.

5 Import the WMF file in the normal manner.

Quick Reference

Commands

IMPORT

Imports files of different formats into the current drawing.

WMFIN

Imports a Windows metafile.

WMFOPTS

Sets options for WMFIN.

Attach Files as Underlays

You can display DWF, DWFx, PDF, and DGN files as underlays in your drawing.

Overview of Underlays

You can underlay and snap to 2D geometry stored in DWF, DWFx, DGN, and PDF files.

Underlays are similar to attached raster images in that they provide visual content but also support object snapping and clipping. Unlike external references, underlays cannot be bound to the drawing.



You can underlay the following file types:

- DWF - A DWF (Design Web Format) file is a highly compressed file format that is created from a DWG file.
- DWFx - DWFx, the future of DWF, is based on the XML Paper Specification (XPS) format from Microsoft.
- PDF - Adobe System's document exchange format.

- DGN - Bentley System's Microstation format. DGN support is limited to V7/V8 DGN files and 2D objects. The 2D objects are imported or attached with full (X, Y, Z) coordinate information, as was present in the original file. If the file contains 3D solids, surfaces, or other 3D objects, a warning is displayed. This content is not supported in this file.

Use the Ribbon Contextual Tab to Work with Underlays

If you select an underlay when the ribbon is active, the DWF Underlay Ribbon Contextual Tab, PDF Underlay Ribbon Contextual tab, or the DGN Underlay Ribbon Contextual tab displays depending on the type of underlay selected. The contextual tab contains options for adjusting, clipping and displaying underlays. The ribbon contextual tab is dismissed automatically after the underlay is deselected.

Attach, Scale, and Detach Underlays

You can add or remove references to underlays within drawing files, or you can change their relative size.

Attach Files as Underlays

You can attach a DWF, DWFx, DGN, or PDF file as an underlay to a drawing file.

You reference and place underlay files in drawing files the same as you do raster image files; they are not actually part of the drawing file. Like raster files, the underlay is linked to the drawing file through a path name. The path to the file can be changed or removed at any time.

NOTE Although underlay files are reproductions of their source drawing, they are not as precise as drawing files. Underlays will show slight discrepancies in precision.

By attaching underlays this way, you can use files in your drawing without greatly increasing the drawing file size. You can only view DWF and PDF underlays in the 2D Wireframe visual style. DGN underlays can be viewed in any visual style.

TIP You can copy selected objects from a DGN underlay file into the current drawing with the NCOPY command.

See also:

- For information on identifying referenced underlays, see *Highlight External References in a Drawing* in Attach and Detach Referenced Drawings on page 1774.

Attach PDF Files

There are a few things specific to PDF files that you do not have to consider with DWF or DGN files. PDF files with more than one page are attached one **page** at a time (as opposed to one **sheet** at a time for DWF files or one **model** at a time for DGN files). Also, hypertext links from PDF files are converted to straight text and digital Signatures are not supported.

Drag Underlay Files

You can drag underlays directly into the drawing. Once you drag the file into the drawing, you can enter

- A sheet name (DWF and DWFx)
- A page number (PDF)
- Or a model name (DGN)

from the file at the Command prompt and then specify an insertion point, scale factor, and rotation.

Attaching an Underlay Multiple Times

You can reattach an underlay multiple times, treating it as a block. Each underlay has its own clip boundary and settings for contrast, fade, and monochrome. However, you cannot bind an underlay to a drawing and you cannot edit or modify the underlay's content.

Layers in Underlay Files

If the underlay file contains layers, you can control how the layers display after attaching the file. If the file does not contain layer information, the Underlay Layers dialog box does not display any layer information.

Underlay Files in Xrefs

DWG file references (xrefs), in a drawing can include an underlay. In this situation, objects in the underlay are visible in the parent DWG file.

For example, drawing A includes a DWF underlay showing some mechanical details. You need the content of drawing A attached to your current drawing, drawing B. If you attach drawing A as an external reference to drawing B, the DWF underlay that was already attached to drawing A is also be present.

All of the property settings made to the underlay in the external reference, such as clipping boundaries, appear as they do in the parent drawing.

Password Protected Underlay Files

If a file you want to attach as an underlay is password protected, you are prompted to enter the password after you have selected the file. You cannot attach the file until you have correctly entered the password.

After an underlay is attached to a DWG file, you are prompted for the underlay file's password each time you open the DWG file. If the DWG file is also password protected, or has several other protected drawing references attached, you may be prompted for multiple passwords.

If you change the path of an underlay to a file that requires a password, you are prompted for a password as well.

NOTE DWF, DWFx, and PDF file passwords are case sensitive but DWG file passwords are not. To make synchronization between DWG and DWF file passwords easier, make sure the DWF, DWFx, and PDF file passwords match the DWG file password and are entirely uppercase.

To attach a DWF or DWFx underlay

1. Click Insert tab ➤ Reference panel ➤ Attach.
2. In the Select Reference File dialog box, select the file you want to attach.
3. Click Open.
4. In the Attach DWF Underlay dialog box, select one sheet, or use SHIFT or CTRL to select multiple sheets.
5. Use one of the following methods to specify the insertion point, scale, or rotation of the underlay file:
   - Select Specify On-Screen to use the pointing device to attach the DWF underlay at the location, scale, or angle you want.
   - Clear Specify On-Screen and enter values under Insertion Point, Scale, or Rotation.
6. Click OK.

To attach a PDF underlay

1. Click Insert tab ➤ Reference panel ➤ Attach.
2. In the Select Reference File dialog box, select the PDF file you want to attach.
3. Click Open.
4. In the Attach PDF Underlay dialog box, select one page, or use SHIFT or CTRL to select multiple pages.
5. Use one of the following methods to specify the insertion point, scale, or rotation of the underlay file:
   - Select Specify On-Screen to use the pointing device to attach the underlay at the location, scale, or angle you want.
   - Clear Specify On-Screen and enter values for Insertion Point, Scale, and Rotation at the command prompt.
6. Click OK.

To attach a DGN underlay

1. Click Insert tab ➤ Reference panel ➤ Attach.
2. In the Select Reference File dialog, select the DGN file you want to attach.
3. Click **Open.**
4. In the Attach DGN Underlay dialog, select a model name and click **OK.**
5. Use one of the following methods to specify the insertion point, scale, or rotation of the underlay file:
   - Select Specify On-Screen to use the pointing device to attach the underlay at the location, scale, or angle you want.
   - Clear Specify On-Screen and enter values for Insertion Point, Scale, and Rotation at the command prompt.
6. Click OK.

Quick Reference

Commands

ATTACH

Inserts an external reference, raster image, or underlay (DWF, DWFx, PDF, or DGN files) in the current drawing.

DGNATTACH

Insert a DGN file as an underlay into the current drawing.

DWFATTACH

Insert a DWF or DWFx file as an underlay into the current drawing.

EXTERNALREFERENCES

Opens the External References palette.

IMAGEATTACH

Inserts a reference to an image file.

NCOPY

Copies objects that are contained in an xref, block, or DGN underlay.

PDFATTACH

Insert a PDF file as an underlay into the current drawing.

XATTACH

Inserts DWG files as an external reference (xref).

System Variables

ERHIGHLIGHT

Controls whether reference names or reference objects are highlighted when their counterparts are selected in the External References palette or in the drawing window.

Cross-reference of MicroStation to AutoCAD Terms

Understanding the differences between MicroStation and AutoCAD terminology helps when attaching DGN files as underlays.

The table below explains many of the different terms that are used in MicroStation and provides a reference to the term that is used in AutoCAD when possible.

MicroStation to AutoCAD Terms

MicroStation Term	AutoCAD Term	Notes
AccuSnap	Osnap	Drafting tool for automatic snaps.
ACS	UCS	Coordinate system acronyms. ACS = Auxiliary Coordinate System UCS = User Coordinate System
ByLevel setting	BYLAYER setting	Setting that controls whether color, line weight, and line style are set for each level (layer).
Cell libraries	N/A	No reference in AutoCAD terminology.
Cells: shared and normal	Blocks	In AutoCAD, all blocks behave like shared cells. There is no reference in AutoCAD terminology to a normal cell.
Design model	Model space	DWG workmode and AutoCAD only allow for a single model. MicroStation DGN workmode allows for multiple models.
DGN file	DWG file	Native file format for each program.
Drop Element	Explode	Command used to demote element/object types to lower level. For example, cells/blocks can be demoted to geometry.
Element Attributes	Properties	Name for characteristics of elements/objects.
Fit View	Zoom extents	Command for zooming in on all elements currently in the drawing.

MicroStation to AutoCAD Terms

MicroStation Term	AutoCAD Term	Notes
		■ MicroStation's "Fit View" tool fits only visible geometry. ■ "Zoom Extents" area includes layers that have display turned off.
Handles	Grips	Vertices on geometry that can be selected and manipulated.
Key-in	Command Line	Place for entering commands/variables manually.
Levels	Layers	Organizational structure used to control the appearance of objects in a drawing.
Line styles	Linetypes	Setting used to control the appearance of the line work in a drawing.
Merge into Master	Bind XREF	Inserting an external reference into the current drawing.
Message Center	Text Window	Place for viewing text messages from program feedback.
Parasolid	ACIS	Native 3D modeling kernel for each program.
Patterning	Hatching	Filling of a defined area with a defined pattern.
Pen tables	Plot styles	Used to control how linework appears when printing or plotting.
References	References: attachments, overlays, and underlays	References to the current drawing are stored externally, which keeps the file size down, but still allows for access to the geometry for drafting and plotting. In Microstation, you can

MicroStation to AutoCAD Terms		
MicroStation Term	**AutoCAD Term**	**Notes**
		have references with Live Nesting or No Nesting. References with No Nesting are translated as overlays in AutoCAD-based products. MicroStation V7 does not support nested references.
Seed file	Template drawing file	Files are used as a starting point for newly created files and store commonly used settings. For DGNEXPORT, the chosen seed file determines what master and sub units are used for translation to DGN as well as resolution and accuracy values.
Sheet model	Drawing layout (paper space)	Commonly used to control the output of a drawing for plotting.
Smart Line	Polyline	Multi-segmented lines.
Tags	Attributes	Element used to store textual information in a cell (block).
View	Viewport	Used to control the visibility of different sections of a drawing.
View Attributes	Drafting settings	There is no direct reference in AutoCAD terminology. Similar options can be found in the taskbar/drafting settings area of AutoCAD.
Working units	Drawing units	Used to control command input values that are dependent on unit values.

Quick Reference

Commands

DGNADJUST

Adjust the fade, contrast, and monochrome settings of a DGN underlay.

DGNATTACH

Insert a DGN file as an underlay into the current drawing.

DGNCLIP

Crops the display of a selected DGN underlay to a specified boundary.

DGNEXPORT

Creates one or more DGN files from the current drawing.

DGNIMPORT

Imports the data from a DGN file into a new DWG file or the current DWG file, depending on the DGNIMPORTMODE system variable.

DGNLAYERS

Controls the display of layers in a DGN underlay.

DGNMAPPING

Allows users to create and edit user-defined DGN mapping setups.

System Variables

DGNFRAME

Determines whether DGN underlay frames are visible or plotted in the current drawing.

DGNIMPORTMAX

Sets the maximum number of elements that are translated when importing a DGN file.

DGNMAPPINGPATH

Specifies the location of the dgnsetups.ini file where DGN mapping setups are stored.

DGNOSNAP

Determines whether object snapping is active for geometry in DGN underlays that are attached to the drawing.

Detach Underlay Files

Underlays that are no longer needed can be detached from a drawing file.

When you detach an underlay, all instances of the underlay are removed from the drawing, and the linking path to the file is removed.

To hide the display of an underlay temporarily, you can unload it rather than detach it. This action preserves the underlay location for reloading later.

NOTE Erasing an individual instance of an underlay is not the same as detaching it. An underlay must be detached to remove the link from your drawing to the file.

To detach an underlay

1. Click Insert tab ➤ Reference ➤  .
2. In the External References palette, in the File References pane, select the underlay you want to detach.
3. Right-click on the underlay and click Detach.

   The underlay is no longer linked to the drawing file. All instances of the underlay are removed from the drawing.

NOTE Erasing an individual instance of an underlay is not the same as detaching it. An underlay must be detached to remove the link from your drawing to the file.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the External References palette.

Unload Underlay Files

To improve performance, you can unload underlays from a drawing file.

When you do not need an underlay in the current drawing session, you can improve performance by temporarily unloading it. Unloaded underlays are not displayed or plotted. Unloading an underlay does not remove its link. If you do not have sufficient memory to open multiple underlays in a drawing, underlays are automatically unloaded.

To unload or reload an underlay

1. Click Insert tab ➤ Reference ➤  .
2. In the File References pane, select a DGN underlay.
3. Right-click, and click Unload or Reload.

   When unloaded, the DGN underlay is still linked to the drawing file, but it is not displayed in the drawing.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the External References palette.

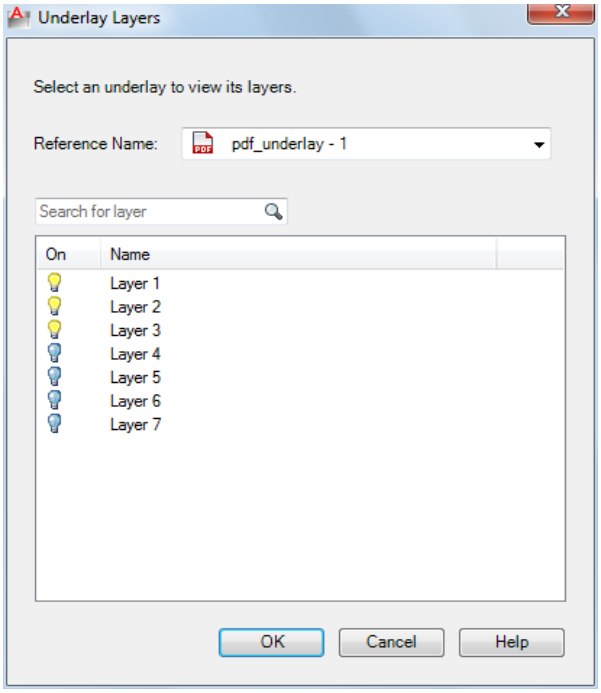
Work with Underlays

You can control the display of layers, use object snaps, and adjust display settings with attached underlays.

Control the Display of Layers in an Underlay

You can turn layers on and off in an underlay.

By default, all visible layers of an underlay are turned on when you attach the file. It is usually convenient to turn off any unneeded layers to reduce the visual complexity of your work.



Use the DWFLAYERS, PDFLAYERS, DGNLAYERS, or ULAYERS command or right-click a selected underlay and click Layers on the shortcut menu.

NOTE If the Underlay Layers dialog box is empty, there are no layers in the underlay.

You can use the Properties palette to determine whether any layers are turned off in an underlay.

- If no layers are turned off, the Layer Display Override property is set to None.
- If at least one layer is turned off, the Layer Display Override property is set to Applied.

To turn layers on and off in an underlay

1. Select an underlay by clicking its bounding frame.
2. Right-click and then do one of the following:
   - If it is a DWF or DWFx underlay, click DWF Layers.

- If it is a PDF underlay, click PDF Layers.
- If it is a DGN underlay, click DGN Layers.

3. In the Underlay Layers dialog box, click the light bulb icon for the layer name that you want to turn on or off.

   You can use CTRL or SHIFT when you click to select several layers.

4. Click Apply to save your changes, or click OK to save and close.

Quick Reference

Commands

DGNLAYERS

Controls the display of layers in a DGN underlay.

DWFLAYERS

Controls the display of layers in a DWF or DWFx underlay.

PDFLAYERS

Controls the display of layers in a PDF underlay.

ULAYERS

Controls the display of layers in a DWF, DWFx, PDF, or DGN underlay.

PROPERTIES

Controls properties of existing objects.

Modify the Position, Scale, or Rotation of an Underlay

You specify an underlay's position, scale, or rotation when you attach an underlay file. You can also modify these settings later during the drafting process.

By default, the insertion point of a file is 0,0,0, its scale factor is 1, and its rotation angle is 0. You can use general modify commands such as move, scale, rotate, mirror, array, and so on. After you select an underlay, you can alter it using the options in the Properties Palette.

TIP Double-clicking an underlay opens the Properties palette for that underlay.

Use Grips with Underlays

While underlay behavior generally mimics raster image behavior, one exception is the way that grips work. In this case, the behavior more closely parallels block behavior. Normally, an underlay displays only a base grip. You can use a base grip to reposition an underlay in a drawing. If you create a clipping boundary, additional grips display for each corner of the boundary. See Clip Underlays on page 1903.

The grip for the base point is the lower-left corner of the underlay.



To edit the position, scale, or rotation of an underlay from the Properties palette

1. In the drawing viewport, double-click the underlay that you want to edit.
2. In the Properties palette, display the Geometry area and do one of the following:
   - To change the underlay position, change the settings for Position X, Y, or Z.
   - To change the underlay scale, width, or height, change the settings for Scale, Width, and Height.
   - To change the underlay rotation, change the setting for Rotation.

To edit the position of an underlay using its base grip

1. Click the frame of an attached underlay.
2. To reposition the underlay, click the base grip.
3. Drag the cursor to the new location and then click again to place the underlay.

Quick Reference

Commands

DWFATTACH

Insert a DWF or DWFx file as an underlay into the current drawing.

DGNATTACH

Insert a DGN file as an underlay into the current drawing.

PDFATTACH

Insert a PDF file as an underlay into the current drawing.

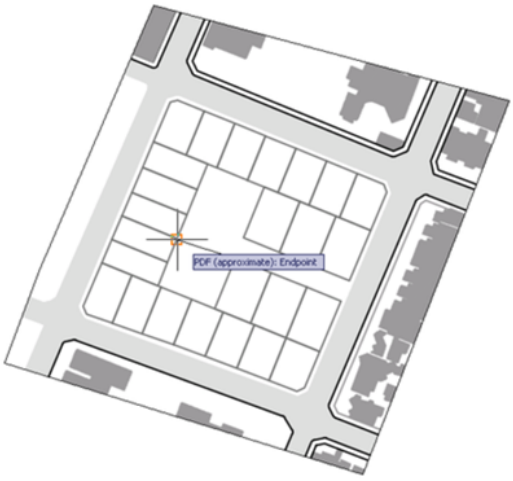
PROPERTIES

Controls properties of existing objects.

Use Object Snaps with Underlays

Use object snaps to draw or edit objects relative to a precise location.

Underlay object snaps are similar to regular object snaps except that they can be turned on and off separately from regular object snaps, and that they apply only to the objects in the attached file.



Object snapping to PDF underlays is similar to object snapping to drawing geometry. However, object snapping might not behave as expected, depending on how the PDF was created. For example, if the PDF was made from *scanned*

architectural plans, the PDF is a raster image, not a vector-based image. Therefore, object snapping does not work. Also, geometry from PDF's that were created outside of AutoCAD may contain nonstandard snapping points, such as circles with no center points.

Use the DWFOSNAP, PDFOSNAP, DGNOSNAP, and UOSNAP system variables to turn object snapping on and off.

Object snapping can also be turned on and off from a shortcut menu. Select an underlay and right-click to display the object snap menu option.

To snap to a geometric point on an underlay

1. Move your cursor over the desired object snap location.

   If UOSNAP is on, your cursor automatically locks onto the snap location in the drawing.

2. Select an object in the drawing.

   The cursor snaps to the eligible location closest to your selection.

Quick Reference

System Variables

DGNOSNAP

Determines whether object snapping is active for geometry in DGN underlays that are attached to the drawing.

DWFOSNAP

Determines whether object snapping is active for geometry in DWF or DWFx underlays that are attached to the drawing.

PDFOSNAP

Determines whether object snapping is active for geometry in PDF underlays that are attached to the drawing.

UOSNAP

Determines whether object snapping is active for geometry in DWF, DWFx, PDF, and DGN underlays that are attached to the drawing.

Adjust Underlay Contrast, Fade, Monochrome, and Colors for the Background

You can modify the contrast, fade, and monochrome settings of an underlay. Also, so that the underlay is visible, you can adjust the underlay colors based on the AutoCAD background colors.

Adjusting these settings does not alter the original file and does not affect other instances of the underlay in the drawing. You can change the contrast, fade, monochrome, and colors in the Properties palette when an underlay is selected. You can also use the DWFADJUST, PDFADJUST, DGNADJUST, or ADJUST commands.

If you change contrast, fade, and monochrome values, plotted output is affected.

Adjust Colors for Background

Adjust Colors for Background controls whether the underlay colors are visible against the drawing background color. The default setting of Yes indicates that the background colors of the underlay and the drawing environment are analyzed to see if they are both light or both dark, or is one dark and the other light. When one background is light and the other dark, the colors of the underlay are inverted so the underlay is displayed. If the setting is changed to No, the original colors of the underlay are used. Depending on the background colors, the underlay might not be visible.

To use the Properties palette to adjust the underlay contrast, fade, monochrome, and background color setting

1. Double-click the underlay to display the Properties palette.
2. In the Properties palette, do one of the following:
   - Adjust the contrast by entering a new value.
   - Adjust the fade by entering a new value.
   - Turn Monochrome On or Off.
   - Set Adjust Colors for Background to Yes or No.

The appearance of the underlay is updated in real-time as you make adjustments in the Properties palette.

To use the DWFADJUST command to adjust the fade, contrast, and monochrome settings of a DWF underlay

1. Click Insert tab ➤ Reference panel ➤ Adjust. 
2. Select the DWF underlay to adjust.
3. On the Enter DWF Underlay Options menu, select one of the following:
   - Select Fade and enter a fade value.
   - Select Contrast and enter a contrast value.
   - Select Monochrome and set monochrome to Yes or No.

The appearance of the DWF underlay is updated after you enter the new value and the command is completed.

To use the PDFADJUST command to adjust the fade, contrast, and monochrome settings of a PDF underlay

1. Click Insert tab ➤ Reference panel ➤ Adjust. 
2. Select the PDF underlay to adjust.
3. On the Enter PDF Underlay Options menu, select one of the following:
   - Select Fade and enter a fade value.
   - Select Contrast and enter a contrast value.
   - Select Monochrome and set monochrome to Yes or No.

The appearance of the PDF underlay is updated after you enter the new value and the command is completed.

To use the DGNADJUST command to adjust the fade, contrast, and monochrome settings of a DGN underlay

1. Click Insert tab ➤ Reference panel ➤ Adjust. 
2. Select the DGN underlay to adjust.

3 On the Enter DGN Underlay Options menu, select one of the following:

- Select Fade and enter a fade value.
- Select Contrast and enter a contrast value.
- Select Monochrome and set monochrome to Yes or No.

The appearance of the PDF underlay is updated after you enter the new value and the command is completed.

Quick Reference

Commands

DWFADJUST

Adjust the fade, contrast, and monochrome settings of a DWF or DWFx underlay.

DGNADJUST

Adjust the fade, contrast, and monochrome settings of a DGN underlay.

PDFADJUST

Adjust the fade, contrast, and monochrome settings of a PDF underlay.

PROPERTIES

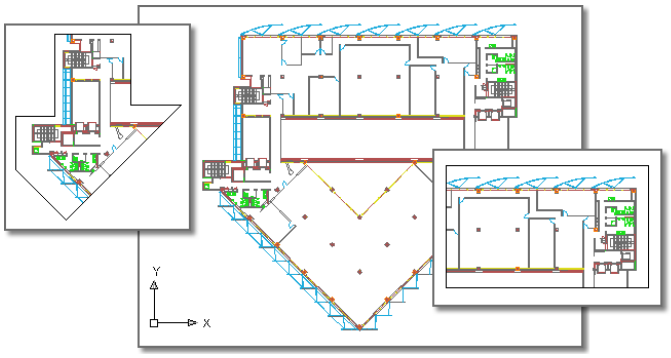
Controls properties of existing objects.

Clip Underlays

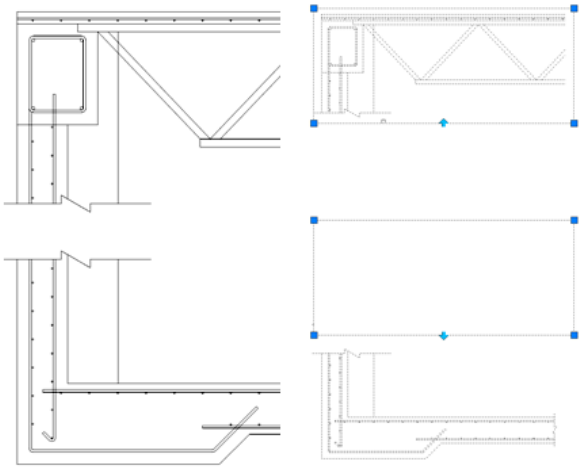
You can use a clipping boundary to clip an underlay.

You can define part of an underlay that you want to display and plot by setting up a clipping boundary with DGNCLIP, DWFCLIP, PDFCLIP, DGNCLIP, IMAGECLIP, VPCLIP, and XCLIP. The clipping boundary can be a closed polyline, rectangle or a polygon with vertices within the overall extents of the underlay. Each instance of an underlay can only have one clipped boundary. Multiple instances of the same underlay can have different boundaries.

Following is an example of an underlay with insets showing polygonal (l) and rectangular (r) clipping boundaries:



When the clipping boundary is no longer needed, you can delete the clipped boundary from the underlay and the underlay is displayed with its original boundary. You can also invert the area to be hidden inside or outside the clipping boundary. With grips located at the midpoint on the first edge of the clipping boundary, you can invert the display of the clipped reference inside or outside the boundary.



You can control the way clipping boundaries and grips display with the clipping frame system variables. The clipping frame system variable are FRAME, PDFFRAME, DGNFRAME, DWFFRAME, XCLIPFRAME, and IMAGEFRAME.

See also:

- Clip External References and Blocks on page 1793
- Clip Raster Images on page 1924

To clip an underlay

1. Click the bounding box of an underlay to select it.
2. Click Contextual tab ➤ Clipping panel ➤ Create Clipping Boundary.



3. At the command prompt, Enter **s** (Select Polyline), **p** (Polygonal), **r** (Rectangular), or **i** (Invert Clip) and then, draw the specified boundary on the underlay.

   To draw a boundary by selecting a polyline, draw or overlay a polygonal shape created with a polyline on the underlay. Then, click Create Clipping Boundary and select the polyline to create a boundary.

   To draw a polygonal boundary, you are prompted to specify consecutive vertices. To finish drawing a polygon, press Enter or right-click anywhere in the drawing area.

To specify a new boundary on a clipped underlay

1. Select a clipped underlay.
2. Click Contextual tab ➤ Clipping panel ➤ Create Clipping Boundary.



3. Enter **y** (Yes) to delete the old boundary.
4. At the command prompt, Enter **s** (Select Polyline), **p** (Polygonal), **r** (Rectangular), or **i** (Invert Clip) and then, draw the specified boundary on the underlay.

   To draw a boundary by selecting a polyline, draw or overlay a polygonal shape created with a polyline on the underlay. Then, click Create Clipping Boundary and select the polyline to create a boundary.

If you are drawing a polygonal boundary, you are prompted to specify consecutive vertices. To finish drawing a polygon, press Enter or right-click anywhere in the drawing area.

To edit the existing boundary on a clipped underlay

1. Select a clipped underlay.
2. Click one of the corner grips and drag it to a new location.
3. Click again to set the corner grip.

For more information, see Modify Objects Using Grips on page 777

To show or hide the clipped portion of an underlay

1. Select the clipped underlay you want to show or hide.
2. Right-click, and click Properties.
3. Scroll to Misc and select Yes or No for the Show Clipped option.

To delete the boundary of a clipped underlay

1. Select the clipped underlay.
2. Click Contextual tab ➤ Clipping panel ➤ Remove Clipping. 

   The boundary of the clipped underlay is deleted, and the original boundary of the underlay is restored.

Quick Reference

Commands

DGNCLIP

Crops the display of a selected DGN underlay to a specified boundary.

DWFCLIP

Crops the display of a selected DWF or DWFx underlay to a specified boundary.

PDFCLIP

Crops the display of a selected PDF underlay to a specified boundary.

PROPERTIES

Controls properties of existing objects.

Hide and Show Underlay Frames

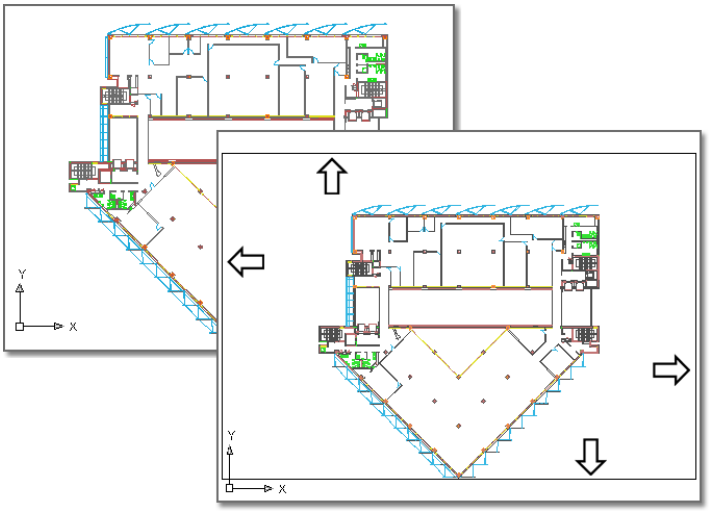
You can display and plot a border around the underlay or the clipping boundary.

A frame is a visual border that shows the extents of the underlay, or the clipped boundary of the underlay. When underlay frames are hidden, clipped underlays are still displayed to their specified boundary limits; only the boundary is affected.

Use the DWFFRAME, PDFFRAME, DGNFRAME, or FRAME system variables to not only display frames, but also to specify whether or not to plot them. The FRAME system variable changes the setting for all underlays in the drawing regardless of type.

NOTE Underlays can be selected if they are not on a locked layer; for example, if the underlay is part of a named selection set made with the All option when selecting objects.

The following foreground example shows the underlay with a visible frame:



The foreground example shows the underlay with a visible frame.

To hide and show underlay frames

Do the following:

1. To hide frames, click Insert tab ➤ Reference panel ➤ Frame Settings drop-down ➤ Hide Frames 
2. To display and plot frames, click Insert tab ➤ Reference panel ➤ Frame Settings drop-down ➤ Display And Plot Frames. 
3. To display frames without plot, click Insert tab ➤ Reference panel ➤ Frame Settings drop-down ➤ Display But Don't Plot Frames. 

Quick Reference

System Variables

DGNFRAME

Determines whether DGN underlay frames are visible or plotted in the current drawing.

DWFFRAME

Determines whether DWF or DWFx underlay frames are visible or plotted in the current drawing.

FRAME

Controls the display of frames for all images, underlays, and clipped xrefs.

FRAMESELECTION

Controls whether the frame of an image, underlay, or clipped xref can be selected.

PDFFRAME

Determines whether the PDF underlay frame is visible.

Manage and Publish Drawings Containing Underlays

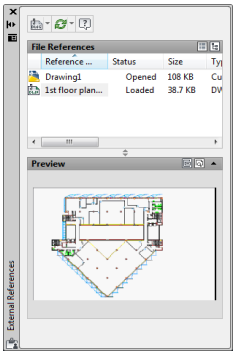
You can view and manipulate underlays and change paths to underlays in the External References palette.

View Underlay Information

You can view file-specific information about the underlays that are attached to a drawing in the External References palette. You can also load and unload underlays and perform other operations there.

In the External References palette, you can view underlay information either as a list or as a tree. To control how the information is displayed in the External References palette, click the List View or Tree View button in the upper-right corner. The list view displays the name of each underlay in the drawing, its loading status, file size, date last modified, and search path. The tree view lists the underlays in a hierarchy that shows their nesting level within xrefs and blocks. The status, size, and other information are not displayed in the tree view.

In either view, you can display information about an underlay; attach or detach the underlay; unload or reload the underlay; and browse for and save a new search path.



Use the List View

The list view displays all underlays attached to the current drawing, but it does not specify the number of instances. It is the default view. You can sort underlays by category by clicking the column heading. Change the width of a column by dragging its border to the right or left.

The following information is displayed in the list view:

- Name of DWF, DWFx, PDF, or DGN file
- Status (loaded, unloaded, or not found)
- File size
- File type
- Date and time file was last saved
- Name of the saved path

If the program cannot find an underlay, its status is listed as Not Found. If the underlay is unreferenced, no instances are attached for the underlay. If the underlay is not loaded, its status is Unloaded. Underlays with a status of Unloaded or Not Found are not displayed in the drawing.

Use the Tree View

The top level of the tree view lists DWF, DWFx, PDF, and DGN files in alphabetical order. In most cases an underlay file is linked directly to the drawing and listed at the top level. However, if an xref or a block contains an attached underlay, additional levels are displayed.

View Underlay File Details

You can preview a selected underlay and view DWF, DWFx, PDF, and DGN file details, including

- Reference name
- Status
- File size
- File type
- File creation date
- Saved path
- Found At path

To display a list of the underlays attached to a drawing

1. Click Insert tab ➤ Reference ➤  .
2. In the External References palette, click the List View button.

To display underlay names and their nesting levels within xrefs and blocks

1. Click Insert tab ➤ Reference ➤  .
2. In the External References palette, click the Tree View button.

To display underlay layer override status

1. Select the underlay of which you want to view the layer override status.
2. Click View tab ➤ Palettes panel ➤ Properties.
3. In the Properties palette, scroll to Misc and view the status of the layer display in the Layer Display Override option.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the External References palette.

DWFLAYERS

Controls the display of layers in a DWF or DWFx underlay.

DGNLAYERS

Controls the display of layers in a DGN underlay.

PDFLAYERS

Controls the display of layers in a PDF underlay.

ULAYERS

Controls the display of layers in a DWF, DWFx, PDF, or DGN underlay.

PROPERTIES

Controls properties of existing objects.

Change File Paths of Underlays

You can change the file path to a referenced DWF, DWFx, PDF, or DGN file or search for an underlay when it is reported as not found.

When you open a drawing with an attached DWF, DWFx, PDF, or DGN file, the path of the selected underlay is displayed in the External References palette under Found At in the Detail list. The displayed path is the actual path where the source file was found. The path where the source file was originally attached is displayed under Saved Path.

To locate the file, the program searches the following paths and folders in the following order:

- Path specified when the underlay was attached
- Folder containing the current drawing file
- Project search paths specified on the Files tab of the Options dialog box
- Support search paths specified on the Files tab of the Options dialog box
- Start In folder specified in the program shortcut

If you open a drawing that contains a DWF, DWFx, PDF, or DGN file that is not in the saved path location or in any of the defined search paths, the External References palette displays **Not Found** in the Status column of the File References list, and the **Found At** entry is blank in the **Details** list.

For more information about using full paths, relative paths, and project names, see Set Paths to Referenced Drawings on page 1782.

To change the underlay path

1. Click Insert tab ➤ Reference ➤  .
2. In the External References palette, select an underlay whose path you want to change. Click the [...] at the right side of the Found At box.
3. In the Select File dialog box, select a new path. Click Open.

   The new path is displayed in the Saved Path column.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the External References palette.

OPTIONS

Customizes the program settings.

System Variables

DGNMAPPINGPATH

Specifies the location of the dgnsetups.ini file where DGN mapping setups are stored.

Publish, Save, and eTransmit Drawings Containing Underlays

When you eTransmit a file, DWF, DWFx, PDF, and DGN underlays are tracked and managed.

Plot and Publish

When a drawing file containing an underlay is plotted or published to a new file, any visible geometry is included in the newly plotted or published file. However, none of the layer data from the original DWF, DWFx, PDF, or DGN attachment gets published with the new file.

Save to a Previous DWG Format

If you save a drawing that contains underlays to a previous DWG format, note the following exceptions:

- DWF underlays do not display and are not replaced by a proxy object in releases earlier than AutoCAD 2007.
- DWFx underlays do not display and are not replaced by a proxy object in releases earlier than AutoCAD 2008.
- PDF underlays are not supported in releases earlier than AutoCAD 2010 (unless you have a Bonus Pack installed).
- DGN underlays are only supported in AutoCAD 2008 or later. They will not display in earlier versions of AutoCAD. Also, if you save a drawing in

AutoCAD 2009, AutoCAD 2008 will only recognize the underlay if it is a V8 MicroStation file.

eTransmit

Underlay attachments are tracked and managed when you use eTransmit in the same way you track and manage raster image attachments.

Quick Reference

Commands

DGNEXPORT

Creates one or more DGN files from the current drawing.

EXPORT

Saves the objects in a drawing to a different file format.

EXPORTDWF

Creates a DWF file and allows you to set individual page setup overrides on a sheet by sheet basis.

EXPORTDWFX

Creates a DWFx file where you can set individual page setup overrides on a sheet-by-sheet basis.

EXPORTPDF

Creates a PDF filewhere you can set individual page setup overrides on a sheet-by-sheet basis.

EXPORTSETTINGS

Adjusts the page setup and drawing selection when exporting to a DWF, DWFx, or PDF file.

PUBLISH

Publishes drawings to DWF, DWFx, and PDF files, or to plotters.

PLOT

Plots a drawing to a plotter, printer, or file.

System Variables

EXPORTEPLOTFORMAT

Sets the default electronic file output type: PDF, DWF, or DWFx.

EXPORTMODELSPACE

Specifies what part of the drawing to export to a DWF, DWFx, or PDF file from Model space

EXPORTPAGESETUP

Specifies whether to export to a DWF, DWFx, or PDF file with the current page setup.

Reference Manager and Underlays

With the Reference Manager, you can view DWF, DWFx, PDF, and DGN underlays that are attached to a DWG file and edit the paths of the attachments.

With the Reference Manager you can check any file dependencies that a drawing file may have. You can also load and unload underlays and perform various other operations. The properties reported by the Reference Manager include

- File type
- Status
- File name
- Reference name
- Saved path
- Found path
- Host drawing
- Host version

You can view the list of references by drawing or by reference type.

NOTE If an underlay's status is Not Found or Unloaded, it displays in the drawing as an underlay boundary box.

To check underlays with the Reference Manager

1. Click Start menu (Windows) ➤ All Programs (Programs) ➤ Autodesk ➤ AutoCAD 2012 ➤ Reference Manager.
2. In the Reference Manager, click the Add Drawings button.
3. In the Add Drawings dialog box, select a drawing and then click Open.

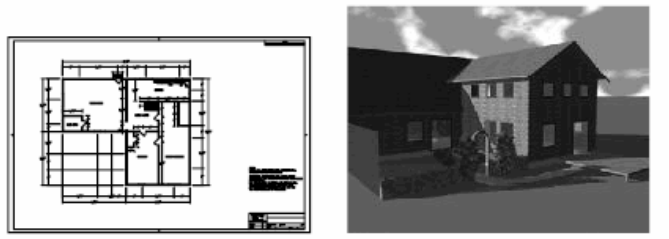
Attach Raster Image Files

You can view and manipulate raster images and associated file paths in drawings.

You can add raster images to your vector-based drawings, and then view and plot the resulting file. There are a number of reasons for combining raster images with vector files, including scanning documents, faxes, or microfilm drawings; using aerial and satellite photographs; using digital photographs; creating effects such as watermarks and logos; and adding computer-rendered images.

Overview of Raster Images

Raster images consist of a rectangular grid of small squares or dots known as *pixels*. For example, a photograph of a house is made up of a series of pixels colorized to represent the appearance of a house. A raster image references the pixels in a specific grid.



Raster images, like many other drawing objects, can be copied, moved, or clipped. You can modify an image with grip modes, adjust an image for contrast, clip the image with a rectangle or polygon, or use an image as a cutting edge for a trim.

The image file formats supported by the program include the most common formats used in major technical imaging application areas: computer graphics,

document management, engineering, mapping, and geographic information systems (GIS). Images can be bitonal, 8-bit gray, 8-bit color, or 24-bit color. Images with 16-bit color depth are not supported.

Several image file formats support images with transparent pixels. When image transparency is set to on, the program recognizes those transparent pixels and allows graphics in the drawing area to "show through" those pixels. (In bitonal images, background pixels are treated as transparent.) Transparent images can be gray-scale or color.

NOTE Although the file name extension is listed in the following table, the file format is determined from the file contents, not from the file extension.

Use the Ribbon Contextual Tab to Work with Raster Images

If you select an image when the ribbon is active, the Image Ribbon Contextual tab displays. The contextual tab contains options for adjusting, clipping and displaying images. The ribbon contextual tab is dismissed automatically after the image is deselected.

Supported image file formats

Type	Description and versions	File extension
BMP	Windows and OS/2 bitmap format	*.bmp*, *.dib*, *.rle*
CALS-I	Mil-R-Raster I	*.gp4*, *.mil*, *.rst*, *.cg4*, *.cal*
DDS	Microsoft DirectDraw Surface	*.dds*
DOQ	USGS Digital Orthophoto Quads	*.doq*
ECW	Enhanced Compression Wavelet	*.ecw*
FLIC	FLIC Autodesk Animator Animation	*.flc*, *.fli*
GeoSPOT	GeoSPOT (BIL files must be accompanied with HDR and PAL files with correlation data, in the same directory)	*.bil*
HDR	High Dynamic Range Image	*.hdr*
IG4	Image Systems Group 4	*.ig4*

Supported image file formats		
Type	**Description and versions**	**File extension**
JPEG2000	Wavelet-based compression standard created by the Joint Photographics Expert Group	*.jp2*, *.j2k*
JFIF or JPEG	Joint Photographics Expert Group	*.jpg*, *.jpeg*
MrSID	Multiresolution Seamless Image Database	*.sid*
NITF	National Imagery Transmission Format **NOTE** NITF files containing elevation data require AutoCAD Raster Design	*.nitf*
OpenEXR	Industrial Light & Magic High-Dynamic Range image	*.exr*
PCX	Picture PC Paintbrush Picture	*.pcx*
PICT	Picture Macintosh Picture	*.pct*
PNG	Portable Network Graphic	*.png*
PSD	Adobe Photoshop Document	*.psd*
RLC	Run-Length Compressed	*.rlc*
TARGA	True Vision Raster-Based Data Format	*.tga*
TIFF	Tagged Image File Format	*.tif*, *.tiff*

Quick Reference

Commands

IMAGE

Displays the External References palette.

IMAGEATTACH

Inserts a reference to an image file.

Attach, Scale, and Detach Raster Images

You can add or remove references to raster images within drawing files, or you can change their relative size.

Attach Raster Images

You can attach a reference to a raster image file to a drawing file using a linked image path. The image file can be accessed from the Internet.

Images can be referenced and placed in drawing files, but like external references (xrefs), they are not actually part of the drawing file. The image is linked to the drawing file through a path name. Linked image paths can be changed or removed at any time.

By attaching images using linked image paths, or by dragging images using DesignCenter™, you can place images in your drawing and only slightly increase the drawing file size. See Add Content with DesignCenter on page 91.

Once you've attached an image, you can reattach it multiple times, treating it as if it were a block. Each insertion has its own clip boundary and its own settings for brightness, contrast, fade, and transparency.

NOTE AutoCAD 2000, AutoCAD LT 2000, and later releases do not support LZW-compressed TIFF files, with the exception of English language versions sold in the US and Canada. If you have TIFF files that were created using LZW compression and want to insert them into a drawing, you must resave the TIFF files with LZW compression disabled.

For information on identifying referenced images, see *Highlight External References in a Drawing* in Attach and Detach Referenced Drawings on page 1774

Access Raster Images Using the Internet

Designers and manufacturers store images of their designs or products on the Internet. You can easily access image files from the Internet. URL image file names are stored in the drawing.

Accessing images from the Internet saves time and provides for rapid distribution of designs. For example, an architect who needs to show a client

what custom cabinets will look like has the manufacturer create a rendered image of the cabinets, post it to a website, and then attach the image to the drawing file as a URL; any design changes can be updated immediately. For more information, see Reference Other Drawing Files on page 1771.

To attach an image

1. Click Insert tab ➤ References panel ➤ Attach. 
2. In the Select Image File dialog box, select a file name from the list or enter the name of the image file in the File Name box. Click Open.
3. In the Image dialog box, use one of the following methods to specify insertion point, scale, or rotation:
   - Select Specify On-Screen to use the pointing device to insert the image at the location, scale, or angle you want.
   - Clear Specify On-Screen and enter values under Insertion Point, Scale, or rotation.

   To view the unit of measurement for the image, click Details.
4. Click OK.

To attach and scale an image file from the Internet

1. Click Insert tab ➤ References panel ➤ Attach. 
2. In the Select Image File dialog box, click the Search the Web button.
3. In the Browse the Web—Open dialog box, enter the URL for the location of the image file in Look In.
4. Enter the image file name in the File Name box. Click Open.

   You can also select an image by right-clicking the image. Click Properties. You can highlight, copy, and paste the complete image URL Address into the File Name box.
5. In the Image dialog box, click OK.
6. If prompted, specify an insertion point and scale factor.

Quick Reference

Commands

IMAGE

Displays the External References palette.

IMAGEATTACH

Inserts a reference to an image file.

Scale Raster Images

You can control the size of a raster image in a drawing to match the scale of the drawing.

You can specify the raster image scale factor when you attach the image so that the scale of the geometry in the image matches the scale of the geometry in the drawing. The default image scale factor is 1, and the default unit for all images is "Unitless." The image file can contain resolution information defining the dots per inch (DPI), relating to how the image was scanned.

If an image has resolution information, the program combines this information with the scale factor and the unit of measurement of the drawing to scale the image in your drawing. For example, if your raster image is a scanned blueprint on which the scale is 1 inch equals 50 feet, or 1:600, and your drawing is set up so that 1 unit represents 1 inch, then in the Image dialog box under Scale, select Specify On-Screen. To scale the image, you clear Specify On-Screen, and then enter **600** in Scale. The image is then attached at a scale that brings the geometry in the image into alignment with the geometry in the drawing.

If no resolution information is defined with the attached image file, the width of the raster image is set to one unit. Thus, when the image file is attached, the image width in units is equal to the raster image scale factor.

To attach and scale an image

1. Click Insert tab ➤ References panel ➤ Attach. 
2. In the Select Image File dialog box, select a file name from the list or enter the name of the image file in the File Name box. Click Open.

3 In the Image dialog box, use one of the following methods to specify insertion point, scale, or rotation:

- Select Specify On-Screen to use the pointing device to insert the image at the location, scale, or angle you want.
- Clear Specify On-Screen and enter values under Insertion Point, Scale, or Rotation.
  To view the unit of measurement for the image, click Details.

4 Click OK.

Quick Reference

Commands

IMAGE

Displays the External References palette.

IMAGEATTACH

Inserts a reference to an image file.

Detach Raster Images

You can detach the reference to an image file in a drawing.

You can detach images that are no longer needed in a drawing. When you detach an image, all instances of the image are removed from the drawing, the image definition is purged, and the link to the image is removed. The image file itself is not affected.

NOTE Erasing an individual instance of an image is not the same as detaching an image. An image must be detached to remove the link from your drawing to the image file.

To detach an image

1 Click Insert menu ➤ External References.

2 In the External References palette, right-click the image name.

3 Click Detach.

The image is no longer linked to the drawing file, and all instances of the image are removed from the drawing.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the External References palette.

Modify Raster Images and Image Boundaries

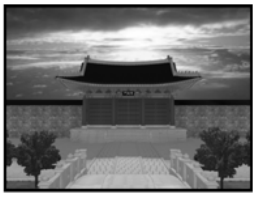
You can control the clipping boundaries and image display properties of a raster image.

Show and Hide Raster Image Boundaries

You can control whether the clipping boundaries of a raster image are displayed or hidden in a drawing.

You can hide image boundaries. Hiding the image boundary prevents the boundary from being plotted or displayed. Also, hiding the image boundary prevents you from selecting the image with the pointing device, ensuring that the image cannot accidentally be moved or modified. However, images can still be selected if they are not on a locked layer, for example, if the image is part of a named selection set made with the All option. When image boundaries are hidden, clipped images are still displayed to their specified boundary limits; only the boundary is affected. Showing and hiding image boundaries affects all images attached to your drawing.





NOTE When an image frame is turned off, you cannot select images using the Pick or Window options of SELECT.

To show and hide image boundaries

1. Click Modify menu ➤ Object ➤ Image ➤ Frame.
2. To hide image boundaries, enter **0**; to show and plot image boundaries, enter **1**; to show image boundaries but not plot them, enter **2**.

Quick Reference

Commands

IMAGEFRAME

Controls whether image frames are displayed and plotted.

System Variables

FRAMESELECTION

Controls whether the frame of an image, underlay, or clipped xref can be selected.

Clip Raster Images

You can clip and display specific portions of a raster image in a drawing with a clipping boundary.

With a clipping boundary, only the parts of the image that you want visible are displayed. You can define the part of an image that you want to display and plot by clipping the image with IMAGECLIP. The clipping boundary can be a polyline, rectangle, or a polygon with vertices within the boundaries of the image. You can change the boundary of a clipped image.You can also delete the clipped boundary of an image. When you delete a clipping boundary, the original image is displayed.





You can invert the area to be hidden, inside or outside the clipping boundary. With grips located at the midpoint on the first edge of the clipping boundary, you can invert the display of the clipped reference inside or outside the boundary.



With IMAGEFRAME system variable, you can control the visibility of the clipping boundary.

See also:

- Clip External References and Blocks on page 1793
- Clip Underlays on page 1903

To clip an image

1. Click Modify menu ➤ Clip ➤ Image.
2. Select the image to clip by selecting the image boundary.
3. At the Command prompt, Enter **n** (New Boundary).
4. Enter **p** (Polygonal) or **r** (Rectangular), and then draw the boundary on the image.

   If you are drawing a polygonal boundary, you are prompted to specify consecutive vertices. To finish drawing a polygon, press Enter or right-click anywhere in the drawing area.

To invert the clipped image

1. Set IMAGEFRAME, to 1. This displays the clipping boundaries.
2. Select the clipped boundary. You should see the grip at the midpoint on the first edge of the clipped boundary.
3. Click the grip to invert the current x-clip region. The grip now points the other way.

To change the boundary of a clipped image

1. Click Modify menu ➤ Clip ➤ Image.
2. Select the image to clip by selecting the image boundary.
3. Enter **n** (New Boundary).
4. Enter **n** (No) or **y** (Yes) to delete the old boundary.
5. Enter **p** (Polygonal) or **r** (Rectangular), and then draw the new boundary on the image.

   If you are drawing a polygonal boundary, you are prompted to specify consecutive vertices. To finish drawing a polygon, press Enter or right-click anywhere in the drawing area.

   You can modify clipped boundaries with grips. See Modify Objects Using Grips on page 777.

To show or hide the clipped portion of an image

1. Select the clipped image you want to show or hide.
2. Right-click in the drawing area. Click Properties.
3. In the Properties palette, select Yes or No in the Show Image list.

To delete the boundary of a clipped image

1. Click Modify menu ➤ Clip ➤ Image.
2. Select the clipped image with the boundary you want to delete.
3. Enter **d** (Delete).

   The boundary of the clipped image is deleted, and the original boundary of the image is restored.

Quick Reference

Commands

IMAGECLIP

Crops the display of a selected image to a specified boundary.

PROPERTIES

Controls properties of existing objects.

System Variables

IMAGEFRAME

Controls whether image frames are displayed and plotted.

Change Raster Image Brightness, Contrast, and Fade

You can change several display properties of raster images in a drawing for easier viewing or special effects.

You can adjust brightness, contrast, and fade for the display of an image as well as for plotted output without affecting the original raster image file and without affecting other instances of the image in the drawing. Adjust brightness to darken or lighten an image. Adjust contrast to make poor-quality images easier to read. Adjust fade to make drawing geometry easier to see over images and to create a watermark effect in your plotted output.

Bitonal images cannot be adjusted for brightness, contrast, or fade. Images fade to the current screen background when displayed, and they fade to white when plotted.

To adjust brightness, contrast, and fade of an image

1. Click Modify menu ➤ Object ➤ Image ➤ Adjust.
2. Select the image to modify.
3. In the Image Adjust dialog box, to adjust brightness, contrast, and fade, use the appropriate slider or enter a value.

   The default value for both brightness and contrast is 50. You can adjust to a maximum brightness of 100 or to a minimum of 0. The default fade value is 0. You can adjust to a maximum fade of 100.
4. Click OK.

Quick Reference

Commands

IMAGEADJUST

Controls the image display of the brightness, contrast, and fade values of images.

PROPERTIES

Controls properties of existing objects.

Modify Color and Transparency for Bitonal Raster Images

Bitonal images are images that consist only of a foreground color and a background color. You can change the foreground color and turn the transparency of the background color on and off.

Bitonal raster images are images consisting only of a foreground color and a background color. When you attach a bitonal image, the foreground pixels in the image inherit the current settings for color. In addition to the modifications you make to any attached image, you can modify bitonal images by changing the foreground color and by turning on and off the transparency of the background.

NOTE Bitonal images and their boundaries are always the same color.

To change the color and transparency of a bitonal image

1. Select the image to modify.
2. Right-click in the drawing area. Click Properties.
3. In the Properties palette, to change image color, click Color.
4. In the Color drop-down list, select a color or click Select Color to open the Select Color dialog box. In the Select Color dialog box, specify a color. Click OK.
5. To change the background of the selected image to transparent, or to change the background image from transparent to opaque, select Yes or No in the Transparency list.

To change only the transparency of a bitonal image

1. Click Modify menu ➤ Object ➤ Image ➤ Transparency.
2. Select the image to modify.

   To change the background of the selected image to transparent, enter **on**. Enter **off** to change the background to opaque.

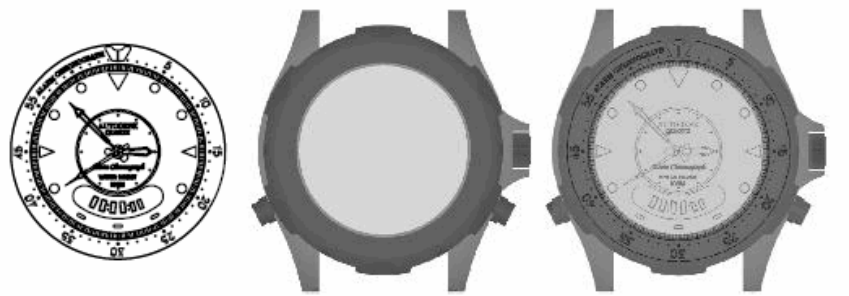


image of a watch face with transparency

conceptual rendering of the watch case constructed and rendered in 3D Studio

result after importing the face layout into AutoCAD, making it transparent, and then placing it over the rendering

Quick Reference

Commands

PROPERTIES

Controls properties of existing objects.

TRANSPARENCY

Controls whether background pixels in an image are transparent or opaque.

Manage Raster Images

You can view and manipulate raster images and change paths to image files using the External References palette.

View Raster Image Information

You can view file-specific information about the raster images that are attached to a drawing. You can also load and unload the images and perform other operations using the External References palette.

In the External References palette, you can view image information either as a list or as a tree in the File References Pane. To control how the information is displayed in the External References palette, click the List View or Tree View button in the upper-right corner. The list view displays the name of each image in the drawing, its loading status, size, date last modified, and search path. The tree view lists the images in a hierarchy that shows their nesting

level within referenced drawings and blocks. The status, size, and other information is displayed below in the Details pane.

In either view, you can display information about an image; attach or detach the image; unload or reload the image; and browse for and save a new search path.

Use the List View

The list view displays the images and any other external references attached to the current drawing, but it does not specify the number of instances. It is the default view. You can sort the external references by categories by clicking the column headings. Change the width of a column by dragging its border to the right or left.

The following information is displayed in the list view:

- Name of the image or selected external reference
- Status (loaded, unloaded, or not found)
- File size
- File type
- Date and time file was last saved
- Name of the saved path

If an image cannot be found, its status is listed as Not Found. A Not Found image is displayed as an image boundary in the drawing even if the IMAGEFRAME system variable is set to off. If the image is unreferenced, no instances are attached for the image. If the image is not loaded, its status is Unloaded. Images with a status of Unloaded or Not Found are not displayed in the drawing.

Use the Tree View

The top level of the tree view lists referenced files in the order that they were attached. In most cases an image file is linked directly to the drawing and listed at the top level. However, if a DWG file reference or a block contains a linked image, additional levels are displayed.

View Image File Details

In the lower panes of the External References palette, you can preview a selected image or view image file details, including

- Image name
- Saved path
- Active path (where the image is found)
- File creation date
- File size
- File type
- Color
- Color depth
- Image size (pixel width and height, resolution and default size)

View Image Information in the Text Window

You can view image information at the Command prompt. Command prompt image information includes image name, image path, the number of definitions, and the number of instances of the image attached to the drawing.

To display a list of the images attached to the drawing

1. Click Insert tab ➤ Reference ➤  .
2. In the External References palette, click the List View button.

To display image names and their nesting levels within DWG file references and blocks

1. Click Insert menu ➤ External References.
2. In the External References palette, click the Tree View button.

To preview an image and view file details

1. Click Insert menu ➤ External References.
2. In the External References palette, select an image file.

3 If the View Pane is not open, click the Preview button.
   The file preview is displayed.

4 To view file details, click the Details button.
   The file details (image size) are displayed.

To preview an image that has not been attached

1 Click Insert menu ➤ Raster Image Referencemage. 

2 In the Select Image File dialog box, select an image file to display a preview.
   If a preview of the image is not displayed, click Show Preview.

3 Click Open.

4 In the Image dialog box, click Details.
   The file details of the image are displayed.

To view image information in the text window

1 At the Command prompt, enter **-image**.

2 Enter **?** (List).

3 Press Enter to list all images.
   The text window displays image information as a list.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the External References palette.

IMAGEATTACH

Inserts a reference to an image file.

Assign Descriptive Names to Raster Images

When the name of a raster image file is not sufficient to identify an image, you can add a descriptive name using the External Reference palette.

Image names are not necessarily the same as image file names. When you attach an image to a drawing, the program uses the file name without the file extension as the image name. Image names are stored in a symbol table; thus you can change the image name without affecting the name of the file. Up to 255 characters are accepted for image file names. In addition to letters and numbers, names can have spaces and any special characters not used by Microsoft® Windows® or AutoCAD for other purposes.

If you attach and place images with the same name but from two different directories, numbers are appended to the image names.

To change an image name

1. Click Insert tab ➤ Reference ➤  .
2. In the External Reference palette, in the Detail Pane, select the image name, and then modify the name.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the External References palette.

Change File Paths of Raster Images

With the External References palette, you can change the file path to a referenced raster image file or search for a referenced image when it is reported as not found.

When you open a drawing with an attached image, the path of the selected image is displayed in the External References palette in the Details Pane under Found At. The path displayed is the actual path where the image file was found. The path where the image file was originally attached is displayed under Saved Path.

To locate the image file, the program searches the following paths and folders in the following order:

- Path specified when the image was attached
- Folder containing the current drawing file
- Project search paths specified on the Files tab of the Options dialog box
- Support search paths specified on the Files tab of the Options dialog box
- Start In folder specified in the program shortcut

If you open a drawing that contains an image that is not in the saved path location or in any of the defined search paths, the External References palette displays Not Found in the image list, and the Found At properties is blank.

You can remove the path from the file name or specify a relative path by editing the path in the Found At properties and then clicking OK in the Select Image File dialog box.

For more information about using full paths, relative paths, and project names, see Set Paths to Referenced Drawings on page 1782.

To change the image path

1. Click Insert tab ➤ Reference ➤  .
2. In the External References palette, select an image whose path you want to change.
3. In the Details Pane, click the Browse button for Found At.
4. In the Select Image File dialog box, browse to the new path. Click OK.

   The new path is displayed in the Found At properties.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the External References palette.

OPTIONS

Customizes the program settings.

System Variables

PROJECTNAME

Assigns a project name to the current drawing.

Tune Raster Image Performance

You can reduce the demands on system performance when manipulating large or many small raster images.

Load and Unload Raster Images

You can improve performance by unloading images when you do not need them in the current drawing session. Unloaded images are not displayed or plotted; only the image boundary is displayed. Unloading an image does not alter its link. If memory is not sufficient to open multiple attached images in a drawing, images are automatically unloaded.

In the External References palette, you can use Reload to reload an unloaded image or to update a loaded image by reloading the image from the specified directory path. If a drawing is closed after an image is unloaded, the image file is not loaded when the drawing is next opened; you must reload it.

To unload or reload images

1. Click Insert tab ➤ Reference ➤  .
2. In the External References palette, in the File References pane, right-click the image name, and then click Unload or Reload.

   The status of the selected image changes.

   All instances of the selected attached images are unloaded or reloaded.

To display specific image instances

1. Select the images you don't want to display.
2. Right-click in the drawing area. Click Properties.
3. In the Properties palette, select Yes or No in the Show Image list.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the External References palette.

PROPERTIES

Controls properties of existing objects.

Improve the Display Speed of Raster Images

To increase the display speed of images, you can change image display quality, hide images not currently needed, use image tiling, or suppress image selection highlighting.

To increase the display speed of images, you can change image display quality from the default high quality to draft quality. Draft-quality images appear more grainy (depending on the image file type), but they are displayed more quickly than high-quality images. Use the IMAGEQUALITY system variable to control image quality.

You can improve the image quality when using True Color (24 or 32 bits per pixel) for raster images by selecting or clearing certain options on the Display tab in the Options dialog box. When images are displayed at optimum quality, regeneration time increases significantly. To improve performance, decrease the number of colors for the system display setting while working in a drawing.

You can increase redrawing speed by hiding images you do not need in the current drawing session. Hidden images are not displayed or plotted; only the drawing boundary is displayed. You can choose to hide an image regardless of the user coordinate system (UCS) in the current viewport.

Use Tiled Images

Tiled images are small portions (a series of tiles) of large images that load much faster than non-tiled images. If you edit or change any properties of an image, only the modified portion is regenerated, thus improving the regeneration time. TIFF (Tagged Image File Format) is the only tiled format that the program supports. The TIFF reader supports all image types:

- Bitonal (1 bit per pixel)
- Gray scale and indexed color (8 bits per pixel)

- True Color (24 or 32 bits per pixel)

You can save tiled TIFF images with most image scanning tools. The image tiles should be no smaller than 64 x 64 pixels and no larger than 512 x 512 pixels. Additional file readers that support other tiled formats, such as CALS Type II, are available from third-party developers.

Suppress Highlighting When Selecting Images

You can turn on or off the highlighting that identifies the selection of a raster image or the image frame by selecting Highlight Raster Image Frame Only on the Display tab in the Options dialog box. You can also set the IMAGEHLT system variable directly. By default, IMAGEHLT is set to 0, to highlight only the raster image frame. Turning off highlighting of the entire image improves performance.

To change the image display quality

1. Click Modify menu ➤ Object ➤ Image ➤ Quality.
2. Enter **d** (Draft) or **h** (High).

   Images are displayed at the specified quality.

To show or hide an image

1. Select the image to modify.
2. Right-click in the drawing area. Click Properties.
3. In the Properties palette, to show or hide the image, select Yes or No in the Show Image list.
4. To switch the background of the selected image between transparent and opaque, select Yes or No in the Transparency list.

Quick Reference

Commands

IMAGEQUALITY

Controls the display quality of images.

OPTIONS

Customizes the program settings.

PROPERTIES

Controls properties of existing objects.

System Variables

IMAGEHLT

Controls whether the entire raster image or only the raster image frame is highlighted.

Export Drawings to Other File Formats

If you need to use the information from a drawing file in another application, you can convert it to a specific format by exporting it.

Export DWF and DWFx Files

Sets of drawings are the primary deliverable for most design groups. Creating a drawing set to distribute for review can be complicated and time consuming. Electronic drawing sets are saved as DWF or DWFx files. You can view or plot DWF and DWFx files using Autodesk Design Review.

To output a single drawing, use the Plot dialog box.

See also:

- Export PDF Files on page 1940
- Review and Markup Files with Design Review on page 2119

To export a drawing to a DWF file

1. Click Output tab ➤ Export to DWF/PDF panel ➤ Export ➤ DWF.
2. In the Save as DWF dialog box, select the desired options, enter a file name, and click **Save**.

To export a drawing to a DWFx file

1. Click Output tab ➤ Export to DWF/PDF panel ➤ Export ➤ DWFx.
2. In the Save as DWFx dialog box, select the desired options, enter a file name, and click **Save**.

To export multiple layouts to a DWF or DWFx file

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 DWF or DWFx from the **Publish To:** drop-down list and click **Publish**.

Quick Reference

Commands

DGNEXPORT

Creates one or more DGN files from the current drawing.

EXPORT

Saves the objects in a drawing to a different file format.

EXPORTDWF

Creates a DWF file and allows you to set individual page setup overrides on a sheet by sheet basis.

EXPORTDWFX

Creates a DWFx file where you can set individual page setup overrides on a sheet-by-sheet basis.

EXPORTPDF

Creates a PDF filewhere you can set individual page setup overrides on a sheet-by-sheet basis.

EXPORTSETTINGS

Adjusts the page setup and drawing selection when exporting to a DWF, DWFx, or PDF file.

PUBLISH

Publishes drawings to DWF, DWFx, and PDF files, or to plotters.

PLOT

Plots a drawing to a plotter, printer, or file.

System Variables

DWFFORMAT

Sets the default format to DWF or DWFx for output in specific commands.

EXPORTEPLOTFORMAT

Sets the default electronic file output type: PDF, DWF, or DWFx.

EXPORTMODELSPACE

Specifies what part of the drawing to export to a DWF, DWFx, or PDF file from Model space

EXPORTPAGESETUP

Specifies whether to export to a DWF, DWFx, or PDF file with the current page setup.

Export PDF Files

You can export a drawing as a PDF file to facilitate sharing information with other design groups.

Sets of drawings are the primary deliverable for most design groups. Creating a drawing set to distribute for review can be complicated and time consuming. Electronic drawing sets are saved as PDF files.

To output a single drawing as a PDF, use the Plot dialog box.

See also:

- Export DWF and DWFx Files on page 1938
- Review and Markup Files with Design Review on page 2119

To export a drawing to a PDF file

1. Click Output tab ➤ Export to DWF/PDF panel ➤ Export ➤ PDF.
2. In the Save as PDF Dialog BoxSave as PDF dialog box, select the desired options, enter a file name, and click **Save**.

To export 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 PDF from the **Publish To:** drop-down list and click **Publish**.

Quick Reference

Commands

EXPORT

Saves the objects in a drawing to a different file format.

EXPORTPDF

Creates a PDF filewhere you can set individual page setup overrides on a sheet-by-sheet basis.

EXPORTSETTINGS

Adjusts the page setup and drawing selection when exporting to a DWF, DWFx, or PDF file.

PUBLISH

Publishes drawings to DWF, DWFx, and PDF files, or to plotters.

PLOT

Plots a drawing to a plotter, printer, or file.

System Variables

EXPORTEPLOTFORMAT

Sets the default electronic file output type: PDF, DWF, or DWFx.

EXPORTPAGESETUP

Specifies whether to export to a DWF, DWFx, or PDF file with the current page setup.

Export DXF Files

You can export a drawing as a DXF file, which contains drawing information that can be read by other CAD systems.

You can export a drawing as a DXF (drawing interchange format) file. DXF files are text or binary files that contain drawing information that can be read by other CAD programs. If you are working with consultants who use a CAD

program that accepts DXF files, you can share a drawing by saving it as a DXF file.

You can control floating-point precision of the DXF format up to 16 decimal places and save the drawing in either ASCII or binary format. ASCII format results in a text file that you can read and edit; binary format results in a significantly smaller file that is faster to work with.

If you do not want to save the entire drawing, you can choose to export selected objects only. You can use this option to remove extraneous material from drawing files.

To create a DXF file

1. Click File menu ➤ Save As.
2. In the Save Drawing As dialog box, in the Files of type box, select a DXF format.
   - To save the entire drawing in DXF format, go directly to step 4.
   - To save specific objects in DXF format, click Tools ➤ Options.In the Saveas Options dialog box, select the DXF format you want and select Select Objects. Enter a value for the number of decimal places of accuracy. Click OK.
3. In the Save Drawing As dialog box, select a folder and enter a file name. Click Save.

   The *.dxf* extension is appended to the file name.
4. If you chose to save specific objects, select the objects and press Enter.

Quick Reference

Commands

SAVEAS

Saves a copy of the current drawing under a new file name.

Export FBX Files

FBX files allow you to import and export 3D objects, 2D objects with thickness, lights, cameras, and materials from one Autodesk program to another.

The FBX file format is an open framework for 3D data transfer that creates a high level of interoperability among Autodesk programs. For example, in Autodesk 3ds Max, you can export a file as an FBX file, and then open that file in AutoCAD to view and edit the objects, lights, cameras, and materials. Likewise, in AutoCAD, you can export a drawing as an FBX file, and then view and edit that file in 3ds Max.

Points (not point clouds), meshes, NURBS curves and surfaces, subdivided meshes, target cameras, lights, and materials are imported into AutoCAD in the same form in which they existed when exported from 3ds Max.

When importing FBX files from 3ds Max into AutoCAD,

- Target cameras created in 3ds Max are imported accurately.
- The LIGHTINGUNITS system variable controls whether lights are imported as standard or photometric lights.

When exporting FBX files from AutoCAD to 3ds Max,

- 3D objects, 2D objects with thickness, lights, cameras, and materials are rotated by 90 degrees on the Z axis.
- You can export 2D objects by increasing their thickness. Any visible and renderable object with thickness can be exported to FBX.

To export a 2D object to an FBX file, increase its thickness to convert to a 3D object. Any visible and renderable object or geometry with thickness can be exported to FBX.

To export a drawing as an FBX file

1. In your drawing, select the objects, lights, cameras, and materials to export to FBX.
2. Click Application menu ➤ Export ➤ FBX.
3. In the FBX Export dialog box, specify a name and location for the file and click Save.

   The FBX Export Options dialog box displays.
4. Specify the items to export, the texture options, and click OK.

Quick Reference

Command

FBXEXPORT

Creates an Autodesk® FBX file containing selected objects in the current drawing.

Export MicroStation DGN Files

You can export DWG files created by AutoCAD-based products to the MicroStation® DGN drawing file format.

The export process translates basic DWG file data into the corresponding DGN file data. There are several translation options to determine how certain data such as external references are handled, the dynamic range of the DGN file as controlled by the master and subunits specified, and whether the DGN file is to accept data with non-zero Z coordinates.

Exchanging and reusing basic drawing data is very useful in collaborative projects. For example, service organizations such as AEC and design-build firms using an AutoCAD-based product might need to export a site plan to DGN format for access by a landscape firm or perhaps a city planning department.

A Standard (default) mapping translation is used to map DWG-format layers, linetypes, lineweights, and colors to DGN-format levels, linestyles, lineweights, and colors. You can remap these elements based on your company CAD standards, therefore, streamlining the export process while minimizing the need for editing the resulting DGN file (see DGNMAPPING).

Understand the Limitations

The DGN import and export capabilities are designed to provide a fundamental exchange of information between MicroStation DGN files and AutoCAD DWG files. However, translating data from one format to a completely different format inevitably requires compromises and substitutions.

- Simple geometric objects such as lines, arcs, and circles, and properties such as layer assignments correlate directly between the DWG and DGN data formats.
- Data with built-in features or variations are visually approximated. For example, text and dimensions might have specialized formatting, and color definitions might be customized.

- Some data cannot be translated completely. For example, product-specific features such as dynamic blocks, can be represented visually but not behaviorally.
- Nested references are not merged into the host DGN file when exporting a DWG file with nested references as V7 DGN. The nested reference is not displayed in the host DGN file but the reference chain still exists.
- AEC objects are not supported and users must use EXPORTTOAUTOCAD command to convert the objects to base ACAD objects before exporting to DGN.
- AutoCAD does not support exporting linetype definitions with DGN files. If a DGN file contains linetype definitions, but .rsc files do not contain these linetype definitions in the Symbology path in MicroStation, then when the DGN file is exported from AutoCAD and opened in MicroStation, the linetypes do not display correctly because of missing linetype definitions.

See the DGNEXPORT Conversion Table specific details about the scope of exporting DWG objects and other data to DGN files.

NOTE Do not export drawings that include V8 DGN underlays to a V7 DGN format file. The V8 DGN references cannot display in a V7 DGN file because the V8 DGN file format is not a valid design file for the resulting V7 DGN file.

Recommendations

To optimize the data transfer to the MicroStation DGN file format, consider the following recommendations:

- Minimize the use of custom objects and other data unique to AutoCAD-based products
- Use simple linetypes as much as possible
- Use simple hatches and hatch styles
- Use simple gradients and gradient styles
- Use simple dimension styles if possible
- Use TrueType text fonts rather than SHX text fonts
- Work primarily in one format, DWG or DGN, rather than switching back and forth

When exporting DWG data to DGN files, you need to specify the conversion units of the DGN *seed file* (master units or sub-units) correctly based on the drawing units of the DWG file. For example, if the DWG file you are exporting to DGN is drawn in inches, and the DGN seed file you are exporting to has master units set to feet and sub-units set to inches, you need to select sub-units in the Export DGN dialog box. If you select master units instead of sub-units, the scale of the imported DWG file will be incorrect. For example, a 36-inch door would display as a 36-foot door.

WARNING Some programs that work with DGN files do not support extended characters that the Windows operating system considers valid for file names. Thus, it is recommended that you do not use accented or Asian characters in file names when using DGNEXPORT.

To export a MicroStation DGN drawing

1. Click Output tab ➤ Send panel ➤ Export. 
2. In the Export Data dialog box, in the Files of type box, select V8 DGN (*.dgn) or V7 DGN (*.dgn).
3. Specify the name of the DGN file at File Name. Do not use accented or Asian characters.
4. Click Save.
5. In the Export DGN Settings dialog box, specify the desired export options.
6. Click OK.

Quick Reference

Commands

DGNEXPORT

Creates one or more DGN files from the current drawing.

Export IGES files

You can export selected objects to a new IGES (*.igs or *.iges) file, which can be read by other CAD systems.

Initial Graphics Exchange Specification (IGES) is a neutral file format designed to transfer 2D and 3D drawing data between dissimilar CAD systems. The IGES standard defines two file formats: fixed-length ASCII, which stores information in 80-character records, and compressed ASCII. The IGESEXPORT and EXPORT commands support only the fixed-length format.

After you start the IGESEXPORT command, you specify the name of the IGES file to create. Thereafter you select what objects to export. Depending on the number of objects you select and their complexity, the export process may take time to complete. If processing takes longer than 5 seconds, IGESEXPORT starts running in the background and displays the Export icon on the status bar.

Once processing is complete, a notification bubble is displayed on the Export status bar. The notification bubble displays the path to the file IGESEXPORT created.

Understanding the Limitations

Under ideal conditions, translation preserves the appearance and functionality of entities. However, this process has limitations, and roundtripping may not be symmetrical.

For example, when exporting to IGES, a 2D polyline is translated as IGES entity 106:12. When importing from IGES, entity 106:12 translates to a spline. Hence, if you use IGESIMPORT to read a file created by IGESEXPORT, the resulting drawing may not be identical to the original drawing.

Similar data loss may occur when transferring data between AutoCAD and another CAD system, using IGES. Usually, a one-time translation to or from IGES does not pose a serious problem. However, data loss can become a concern if drawings are repeatedly translated in and out of IGES.

Recommendations

Inspect the conversion tables of the IGES translators of both the target CAD program and AutoCAD. Get to know what entities are supported. Restructure the drawing to minimize the usage of objects that require compromises and substitutions.

See the conversion table for IGESEXPORT for specific details of the scope of the IGES translator.

To export selected objects to an IGES file

1. Click Output tab ➤ Send panel ➤ Export. 
2. In the IGES Export Data dialog box, in the Files of type box, select IGES (*.igs, *.iges).
3. In the File Name box, specify the name of the IGES file.
4. Click Save.
5. Select the objects to export and press ENTER to begin the export process.

   When export is finished, a notification bubble is displayed.

Quick Reference

Commands

IGESEXPORT

Saves selected objects in the current drawing to a new IGES (*.igs, *.iges) file.

EXPORT

Saves the objects in a drawing to a different file format.

Export WMF Files

You can export selected objects to a WMF (Windows metafile format) file.

While WMF files can contain both vector and raster data, AutoCAD-based products export only vector data.

To create a Windows metafile (WMF)

1. Click Output tab ➤ Send panel ➤ Export. 
2. In the Export Data dialog box, in the Files of type box, select Metafile (*.wmf).
3. Select a folder and enter a file name. Click Save.
4. Select the objects to be included in the exported file.

Quick Reference

Commands

EXPORT

Saves the objects in a drawing to a different file format.

WMFOUT

Saves objects to a Windows metafile.

System Variables

WMFBKGND

Controls the background display when objects are inserted in Windows metafile (WMF) format.

Export Raster Files

You can create a device-independent raster image of the objects in your drawing.

Several commands can be used to export objects into device-independent raster images in the bitmap, JPEG, TIFF, and PNG formats.

Objects are displayed in the raster image as they appear on the screen, including objects in shaded and rendered viewports.

File formats such as JPEG are compressed as they are created. Compressed files take up less disk space, but they might not be readable by certain applications.

To create a bitmap file (BMP)

1. At the Command prompt, enter **bmpout**.
2. In the Create Raster File dialog box, select a folder and enter a file name. Click Save.

   The *.bmp* extension is appended to the file name.
3. Select the objects you want to save.

To create a JPEG file (JPG)

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

2. In the Create Raster File dialog box, select a folder and enter a file name. Click Save.

   The *.jpg* extension is appended to the file name.

3. Select the objects you want to save.

To create a TIFF file (TIF)

1. At the Command prompt, enter **tifout**.
2. In the Create Raster File dialog box, select a folder and enter a file name. Click Save.

   The *.tif* extension is appended to the file name.

3. Select the objects you want to save.

To create a Portable Network Graphics file (PNG)

1. At the Command prompt, enter **pngout**.
2. In the Create Raster File dialog box, select a folder and enter a file name. Click OK.

   The *.png* extension is appended to the file name.

3. Select the objects you want to save.

Quick Reference

Commands

BMPOUT

Saves selected objects to a file in device-independent bitmap format.

EXPORT

Saves the objects in a drawing to a different file format.

JPGOUT

Saves selected objects to a file in JPEG file format.

PNGOUT

Saves selected objects to a file in a Portable Network Graphics format.

TIFOUT

Saves selected objects to a file in TIFF file format.

Export PostScript Files

You can convert a drawing file to a PostScript file, a format that is used by many desktop publishing applications.

The PostScript file format type is used by many desktop publishing applications. Its high-resolution print capabilities make it preferable to raster formats, such as GIF, PCX, and TIFF. By converting the drawing to a PostScript format, you can also use PostScript fonts.

Export in PostScript Format

When you export a file in PostScript format as an EPS file, some objects are handled specially.

- **Thickened text, text control codes**. If text has a thickness greater than 0 or contains control codes (such as %%O or %%D), it is not plotted as PostScript text, although the text is accurately plotted. International and special symbols (such as %%213) are output as PostScript text.
- **ISO 8859 Latin/1 character set**. When text uses character codes in the 127 to 255 range, the text is interpreted according to the ISO 8859 Latin/1 character set. If such a character appears in text that is mapped to PostScript, a version of the font is generated with an encoding vector remapped to represent the ISO character set. The resulting text is output in PostScript in a form compatible with the font.
- **Circles, arcs, ellipses, elliptical arcs**. Except when they have thickness, arcs and circles are translated into the equivalent PostScript path objects.
- **Filled solids**. A solid fill is plotted as a PostScript filled path.
- **Two-dimensional polylines**. A 2D (planar) polyline with uniform width is output as a PostScript stroked path. The PostScript end cap and miter limit variables are set to approximate the segment joining.

See also:

- Plot Adobe PostScript Files on page 1723

To export a drawing to an Encapsulated PostScript (EPS) file

1. Click Output tab ➤ Send panel ➤ Export. 
2. In the Export Data dialog box, specify a location and file name for the EPS file.
3. Under Files of type, select Encapsulated PS (*.eps)
4. Click Save.

Quick Reference

Commands

EXPORT

Saves the objects in a drawing to a different file format.

PLOT

Plots a drawing to a plotter, printer, or file.

Export ACIS SAT Files

You can export certain object types to an ACIS file in ASCII (SAT) format.

You can export trimmed NURBS surfaces, regions, and 3D solids to an ACIS file in ASCII (SAT) format. Other objects, such as lines and arcs, are ignored.

Use the ACISOUTVER system variable to specify the ACIS version for the output of the ACISOUT command.

To create an ACIS file

1. Click Output tab ➤ Send panel ➤ Export. 
2. In the Export Data dialog box, enter a file name.
3. Under Files of type, select ACIS (*.sat). Click Save.
4. Select the objects you want to save as an ACIS file.
   The file extension *.sat* is automatically appended.

Quick Reference

Commands

ACISOUT

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

EXPORT

Saves the objects in a drawing to a different file format.

System Variables

ACISOUTVER

Controls the ACIS version of SAT files created using the ACISOUT command.

Export Stereolithography STL Files

You can export 3D solid objects in the STL file format compatible with stereolithography or 3D printing.

Stereolithography or 3D printing is used in rapid prototyping, rapid manufacturing, and other applications to create physical parts and models.

The 3D solid data is translated to a faceted mesh representation consisting of a set of triangles and saved to an STL file. Use the FACETRES system variable to adjust the facet density to an appropriate detail level.

NOTE Setting the resolution too high slows down the fabrication process without improving the output quality of the stereolithography device.

The STL data is used to create an object by depositing a succession of thin layers of plastics, metals, or composite materials. The resulting parts and models are commonly used for the following:

- Visualize design concepts
- Create product mockups, architectural models, and terrain models
- Test form, fit, and function
- Identify design problems
- Create masters for vacuum forming applications
- Create Marketing tools

To export 3D solids to an STL file

1. Click Output tab ➤ Send panel ➤ Export. 
2. In the Export Data dialog box, enter a file name.
3. Under Files of type, select Lithography (*.stl). Click Save.
4. Select one or more solid objects. All objects must be entirely within the positive XYZ octant of the world coordinate system (WCS). That is, their *X*, *Y*, and *Z* coordinates must be greater than zero.

   The file extension *.stl* is automatically appended to the file name.

Quick Reference

Commands

EXPORT

Saves the objects in a drawing to a different file format.

System Variables

FACETRES

Adjusts the smoothness of shaded and rendered objects and objects with hidden lines removed.

Use Drawings from Different Versions and Applications

You can share drawing files from AutoCAD and AutoCAD LT, drawing files from previous versions, and drawing files that contain custom objects. In some cases there are limitations.

Work with Drawings in Earlier Releases

When you work with drawings created in AutoCAD 2008 (and later releases) in AutoCAD 2007 (and earlier releases), you should be aware of the following visual fidelity issues.

Visual Fidelity for Annotative Objects in Previous Releases

You can specify that annotative on page 2289 objects maintain visual fidelity when they are viewed in AutoCAD 2007 (and earlier releases) with 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 (SAVEFIDELITY set to 1).

Annotative objects may have multiple scale representations on page 2318. When visual fidelity is on, annotative objects are decomposed and scale representations are saved (in an anonymous block on page 2289) to separate layers. These layers 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 recommended that you do not 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 visual fidelity for annotative objects is not selected, a single model space representation is displayed on the Model tab. Depending on the setting of the ANNOALLVISIBLE system variable, more annotation objects may be displayed on the Model tab, and more objects may be displayed in paper space viewports at different sizes than in AutoCAD 2008 and later releases.

For a procedure to set visual fidelity for annotative objects, see To save drawings with visual fidelity for annotative objects.

Annotative Object Properties in Previous Releases

In an AutoCAD 2008 drawing, when an annotative block does not have its paper orientation set to match the layout, and the block contains multiline attributes that are based on a text style that is not set to match the orientation of the layout, the attributes may shift positions if you open this drawing in AutoCAD 2007 (and earlier releases).

Layer Property Overrides in Previous Releases

When you open an AutoCAD 2008 drawing containing layer property overrides, overrides are not visible. The property override settings are retained when the drawing is saved in a previous release, and are visible again when the drawing is opened in AutoCAD 2008.

If a viewport containing layer property overrides is deleted when the drawing is opened in a previous release, the override settings are not retained and are not available when the drawing is opened in AutoCAD 2008.

When the VISRETAIN system variable is set to 0 when the drawing is opened in a previous release, xref layers containing viewport property overrides are not retained.

If you open an AutoCAD 2008 drawing in a previous release, property overrides may display in a thumbnail image. When the drawing is saved with a layout tab, and then opened in the previous release, those property overrides do not display.

DGN Underlays in Previous Releases

DGN underlays do not display in releases prior to AutoCAD 2008.

Dimension Enhancements in Previous Releases

AutoCAD 2008 dimension enhancements are lost when they are edited in earlier releases. If you don't change these dimensions, they are restored when you open the drawing in AutoCAD 2008.

The following dimension enhancements do not lose visual fidelity in previous releases if they are not edited:

- Dimension breaks
- Jogged linear dimensions
- Inspection dimensions
- Angular dimensions that are dimensioned using the quadrant option
- Arc extension lines for radial and diameter dimensions

Multileader Objects in Previous Releases

Multileaders display as proxy objects in releases prior to AutoCAD 2008. The PROXYSHOW system variable controls the display of proxy objects in a drawing.

MTEXT Paragraph and Paragraph Line Spacing in Previous Releases

Some of the new paragraph spacing and paragraph line spacing options are not supported when an AutoCAD 2008 mtext object is opened in AutoCAD 2007 (and earlier releases).

The following mtext formatting features have *no* visual fidelity in previous releases:

- Paragraphs with justified alignment
- Paragraphs with distributed alignment
- Fields that wrap across columns
- Fields that wrap across lines that have new paragraph alignments
- Paragraphs with non-default alignments in mtext without left object-level justification

The following mtext formatting features have *some* visual fidelity in previous releases (when it's possible to add white spaces or replace text with white spaces):

- Paragraphs with non-default alignments (other than justified or distributed) in mtext that has left object-level justification
- Paragraphs with tabs using new tab alignments (center, right, or decimal alignment applied)
- Paragraphs with new line spacing that can be "approximated" with "tall" spaces

Mtext with new formatting that is edited and saved in previous releases loses the new formatting when re-opened in AutoCAD 2008.

Tables in Previous Releases

Editing AutoCAD 2008 tables in previous releases removes AutoCAD 2008 table formatting. Also, AutoCAD 2008 table cells with long block and text strings may extend outside of cell borders when opened in previous releases.

Data Extraction Tables in Previous Releases

For tables that were created with the Data Extraction wizard in AutoCAD 2008, you can't edit or update the extracted data in previous releases.

Materials in Previous Releases

AutoCAD 2008 introduces new procedural maps for materials and luminance. Drawings created earlier than AutoCAD 2007 that contain materials need to be converted. AutoCAD 2008 provides a system variable, 3DCONVERSIONMODE, that is set to automatically convert old materials to

the AutoCAD 2008 format. However, there are some fidelity issues when saving back to previous versions of AutoCAD.

- Procedural maps Checker, Noise, Speckle, Tiles, and Waves are not available in AutoCAD 2007 (and previous releases.) They do not appear when an AutoCAD 2008 drawing is opened in AutoCAD 2007. Marble and Wood are available but the material mayshow some variation.
- The behavior of the Opacity map channel under the Realistic material type has been corrected in AutoCAD 2008. In AutoCAD 2008, the Opacity map channel behaves like a transparency map, where white is transparent and black is opaque (this was how it behaved prior to AutoCAD 2007). In AutoCAD 2007, white represented opaque areas and black represented transparent areas.
- The spherical map rotation is different in AutoCAD 2008. When you open a drawing prior to AutoCAD 2008, the seam for the map is towards the +Y axis. In AutoCAD 2008, it is towards the +X axis.

Lighting in Previous Releases

AutoCAD 2008 provides two options for lighting: standard (generic) lighting and photometric lighting. Previous versions offered only standard (generic) lighting. There is no explicit conversion required for lights from AutoCAD 2007 to AutoCAD 2008. Drawings from AutoCAD 2007 open by default in AutoCAD 2008 in the standard (generic) lighting workflow. The additional photometric properties available in AutoCAD 2008 are available as soon as the photometric lighting workflow is enabled in the drawing.

There is a conversion process required for drawings with lighting prior to AutoCAD 2007. You can use a system variable, 3DVCONVERSTIONMODE, to automatically convert drawings with lighting from previous versions to the AutoCAD 2007 and AutoCAD 2008 lighting format. 3DCONVERSIONMODE has three settings. If set to 0, no conversion takes place. If set to 1, the default value, the conversion takes place automatically. If set to 2, then you are prompted when lights need to be converted and will have the option to convert or not to convert.

Other lighting fidelity issues include

- **Lights in blocks.** Lights in blocks created in AutoCAD 2008 do not always display in AutoCAD 2007.
- **Texture illumination.** In AutoCAD 2007 (and earlier releases), you can't add lighting to textures. In AutoCAD 2008, you can.

- **Ground Shadows.** The intensity of ground shadows in AutoCAD 2008 depends on the brightness of the light and angle of incidence.

Multiple-Language Support in Previous Releases

Drawing properties in AutoCAD 2008 are saved with Unicode characters. For instance, if you save the latest format drawing containing multiple language drawing properties to a 2004-format drawing, the drawing properties are converted to the native characters of the current Windows language. If text cannot be converted to the native characters, it is saved to CIF codes (\U+nnnn) or MIF codes (\M+nxxxx).

When saving the latest format drawing to a 2004-format drawing, any new symbol or dictionary names (for example, layout name, text style name, dimension style name) created in AutoCAD 2008 are saved in the language that was used when the symbol names were created.

In order to view and edit drawings with characters that are not included in the languages specified for your operating system, make sure supplemental language support is installed in your computer operating system. You can specify the language in the Regional and Language Options dialog box, available from the Windows Control Panel. (You may be able to view text that uses SHX fonts without specifying extra language support.)

Text styles for Asian languages that use SHX and Big Font can support characters only from the same code page. For example, text styles that use a Japanese Big Font cannot support German or Korean characters. (English characters, which are part of every code page, are supported.) Multiple-language support for non-Asian languages is supported for text styles that use SHX fonts with Big Fonts disabled. (The SHX font must define the required characters.)

Multiple-language support does not exist in some earlier versions of AutoCAD. For example, when you save a file to AutoCAD 2000 format, the contents of multiple-language multiline text may be corrupted. This problem is more likely to happen when you open and save a drawing on an operating system with a system language setting that differs from the system in which the drawing was last saved.

NOTE Drawings that include external references (xrefs) to drawing files saved in earlier releases also have the limitations described above.

See also:

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-

To save drawings with visual fidelity for annotative objects

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Open and Save tab, under File Save, select Maintain Visual Fidelity for Annotative Objects.
3. Click OK.

Save Drawings to Previous Drawing File Formats

You can save a drawing in a format compatible with previous versions of the product.

You can save a drawing created with the current release of the program in a format compatible with previous versions. This process creates a drawing with information specific to the current release stripped out or converted to another object type.

If you use the current release to open a drawing created with a previous release, and you do not add any information specific to the current release, you can then save the drawing in the format of the previous release without loss of data.

NOTE To use files with AutoCAD Release 12 or AutoCAD LT Release 2, save the drawing using the AutoCAD R12/LT2 DXF option.

If you need to keep a drawing created in a previous release in its original format, either mark the file as read-only, or open it in the current release and use the File Type options in the Save As dialog box to save it in its original format.

Because saving a drawing in an earlier release format may cause some data loss, be sure to assign a different name to avoid overwriting the current drawing. If you overwrite the current drawing, you can restore the overwritten version from the backup file (*filename.bak*) that is created during the saving process.

Maintain Associativity in Dimensions

Associative dimensions created in AutoCAD 2002 or later generally maintain their associativity when saved to a previous release and then reopened in the current release. However, if you modify dimensioned objects using a previous release to the extent that new objects are formed, the dimension associations change when the drawing is loaded into the current release. For example, if a line that was dimensioned is trimmed so that an interior portion of the line

is removed, two line objects result and the associated dimension applies to only one of the line objects.

Dimension associativity is not maintained when a drawing is saved as an AutoCAD R12/LT 2 DXF file and then reopened in the current release.

Save Drawings with Large Objects

Drawings saved to a legacy drawing file format (AutoCAD 2007 or earlier) do not support objects greater than 256MB. For more information about saving drawings that contain large objects to a previous release, see Maintain Compatibility with Large Object Limits on page 240.

Limitations of Saving to Earlier Versions

Saving a drawing in Release 2000/LT 2000 format is subject to the following limitations:

- File size can increase.
- Encryption and digital signatures are not preserved.

Saving a drawing in Release 14/LT 98/LT 97 format is subject to the following limitations:

- Hyperlinks are converted to Release 14/LT 98/LT 97 attached URLs.
- Database links and freestanding labels are converted to Release 14/LT 98/LT 97 links and displayable attributes.
- Database attached labels are converted to multiline text and leader objects, and their link information is not available. Attached labels are restored if you open the drawing in AutoCAD 2000 or later.
- Dynamic block geometry can be redefined independent of the block's dynamic elements, and the geometry in the block reference is not updated when the drawing is opened in AutoCAD 2012 or later.
- Dimensions created using the DIMARC and DIMJOGGED commands may not retain their original color in Release 14/LT 98/LT 97.

Saving a drawing in Release 12/LT 2 DXF format is subject to the following limitations:

- Lightweight polylines and hatch patterns are converted to Release 12 polylines and hatch patterns.

- All solids, bodies, regions, ellipses, leaders, multilines, rays, tolerances, and xlines are converted to lines, arcs, and circles as appropriate.
- Groups, complex linetypes, OLE objects, and preview images are not displayed.
- Many objects are lost if you save a drawing as Release 12 and open it in Release 2000/LT 2000 or later.
- Multiple layouts and layout names are lost. Only the Model tab and the current layout tab are saved.
- Spaces in the names of layers and other objects are converted to underscores, and their maximum length is 32 characters.
- DWF or DWFx underlay files attached to drawings cannot be saved to Release 12/LT 2 DXF format.
- The status of external references as unloaded is lost.

To save an drawing to a previous release format

1. Click File menu ➤ Save As.
2. In the Save Drawing As dialog box, in the File Name box, enter a new drawing name.
3. Under Files of type, select the AutoCAD or AutoCAD LT file format you want.
4. Click OK.

Quick Reference

Commands

CONVERT

Optimizes 2D polylines and associative hatches created in AutoCAD Release 13 or earlier.

OPTIONS

Customizes the program settings.

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.

System Variables

PLINETYPE

Specifies whether optimized 2D polylines are used.

Work with AutoCAD Drawings in AutoCAD LT

AutoCAD LT offers full compatibility when working with AutoCAD drawings. However, you should understand how AutoCAD LT handles AutoCAD-only features.

Work with Fields

In AutoCAD, you can insert LispVariables fields, which are not available in AutoCAD LT. However, the drawings created in AutoCAD that contain LispVariables can be opened without errors in AutoCAD LT, and the cached value is displayed.

Work with Multiple User Coordinate Systems

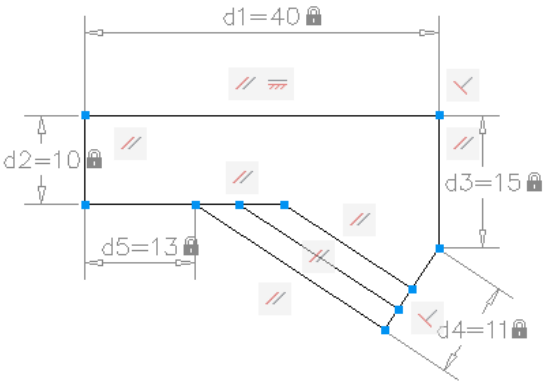
In AutoCAD, you can choose to use a different user coordinate system (UCS) in each viewport in a single drawing file. In AutoCAD LT, you can use only one UCS in each drawing file. The AutoCAD LT behavior is the same as it was in previous releases.

When you open an AutoCAD drawing file in AutoCAD LT, AutoCAD LT uses only the UCS from the current viewport. If you edit the drawing in AutoCAD LT, and then save it and reopen it in AutoCAD, you may notice some discrepancies in UCS usage. User coordinate systems that were set individually in AutoCAD will probably change if the viewports that use them were activated in the AutoCAD LT session.

Work with AutoCAD 2D and 3D Solid Object Shading

In AutoCAD, visual styles provide shading and wireframe options for objects in the current viewport. AutoCAD LT does not support visual styles. The SHADEMODE command in AutoCAD LT provides only the 2D Wireframe and Hidden options. You can use SHADEMODE in AutoCAD LT to turn off visual styles in viewports that were created in AutoCAD. This exposes the underlying geometry so you can easily edit drawings and use the geometry with precision drawing tools such as object snaps.

Work with Constraints



Some of the drawings that you work with will contain design requirements enforced within the drawing itself through the use of constraints. Using constraints, you can enforce requirements while experimenting with different designs.

A constrained object will move in a predictable manner when edited or moved. A single variable change can cause all related objects to change automatically, enabling you to run through design iterations simply and effectively.

There are two general types of constraints supported: Geometric and Dimensional.

- Geometric constraints determine the relationships between 2D geometric objects or points on objects relative to each other.
  Use constraint bars to view the geometric constraints applied to objects. Constraint bars are visible only when you place your cursor over the highlighted nodes.
- Dimensional constraints control distances or angles between 2D geometric objects in a drawing.
  The main dimensional constraints are: dynamic, annotational, and reference constraints.
  - Dynamic constraints (default) - Used to constrain objects and are displayed on demand.
  - Annotational constraints - Used to create associative variables, offset distances, and so on.
  - Reference constraints (read-only) - Read-only dimensional constraints (either dynamic or annotational).

When you place your cursor over a constrained object, you will see a glyph denoting the object is constrained.

With AutoCAD LT, you can do the following:

- View drawings containing constraints created using AutoCAD.
- View and edit the geometric and dimensional constraints.

NOTE You cannot create constraints within AutoCAD LT.

Work with Dynamic Blocks

In AutoCAD, you can add new constraint parameters to a dynamic block in the Block Editor. In AutoCAD LT, you can open drawings which include a block definition containing geometric or dimensional constraints or a block properties table but cannot add new parameters to the block definition.

When you work with constrained dynamic blocks in the Block Editor in AutoCAD LT, you can modify the constraint values and delete constraints but cannot add constraints in the Block Editor. You can manage the constraints while in the Block Editor from the Parametric tab.

Modify 3D Point Clouds

While you cannot index or attach a point cloud in AutoCAD LT, you can open drawings created in AutoCAD that contain point clouds. When an attached point cloud file is unlocked, you can select the point cloud with a grip that is displayed at the centroid of the point cloud.

NOTE You cannot explode a point cloud.

Quick Reference

System Variables

FIELDDISPLAY

Controls whether fields are displayed with a gray background.

PERSPECTIVE

Specifies whether the current viewport displays a perspective view.

POINTCLOUDAUTOUPDATE

Controls whether a point cloud is regenerated automatically after manipulation, panning, zooming, or orbiting.

POINTCLOUDDENSITY

Controls the number of points displayed at once for all point clouds in the drawing view.

POINTCLOUDRTDENSITY

Improves performance by degrading the number of points displayed in the drawing view zooming, panning, or orbiting in real time.

Work with Custom and Proxy Objects

Custom objects provide additional capabilities to the program and related products. When the application that created the custom object is not available, a proxy object is substituted in its place.

A custom object is a type of object created by an ObjectARX® (AutoCAD Run-Time Extension) application, which typically has more specialized capabilities than standard AutoCAD objects. Custom objects include parametric solids (AutoCAD® Mechanical), intelligently interactive door symbols (AutoCAD® Architecture), polygon objects (AutoCAD® Map 3D), and associative dimension objects (AutoCAD and AutoCAD LT).

In addition to Autodesk, many software vendors use ObjectARX to write programs that create graphical and nongraphical custom objects that are useful in their AutoCAD based applications.

Proxy Objects

A proxy object is a substitute for a custom object when the ObjectARX application that created the custom object is not available to AutoCAD or other host applications. Later, when the application is available, the proxy object is replaced by the custom object.

Proxy objects have significantly reduced capabilities compared to their corresponding custom objects. The extent to which proxy objects can be edited is determined by the parent ObjectARX application. For example, operations such as erasing and moving an object, or changing object properties, may or may not be possible on a proxy object, depending on the application that created it.

When you open a drawing, you might see a Proxy Information dialog box. The dialog box tells you the total number of proxy objects in the drawing (both graphical and nongraphical) and the name of the missing application and provides additional information about the proxy object type and display state. You can use the dialog box to control the display of proxy objects.

Object Enablers

An object enabler is a tool that provides specific viewing and standard editing access to a custom object in the host applications when the application that created the custom object is not present.

Object Enablers allow custom objects in a drawing to behave with more intelligence than proxy graphics. Object enablers also facilitate workgroup collaboration when using other Autodesk products.

If the ObjectARX application is not installed on your system, you can check for available Object Enablers on the Web. For example, if you receive a drawing that contains objects that were created in AutoCAD Architecture, but you don't have that application installed on your system, the AEC Object Enabler is downloaded so you can view those drawings as they were intended.

To control whether the program checks for Object Enablers, use the Live Enabler settings on the System tab of the Options dialog box.

For a complete list of the currently available Object Enablers, go to the Autodesk Web site at *http://www.autodesk.com/enablers*.

Object Classification

If an application such as AutoCAD® Map 3D was used to add a feature (object) classification to an object, you can view the classification in the Properties palette in the Class Name entry. If the object's classification is missing from the associated classification (XML) file, or if the associated classification file is missing, an exclamation mark is displayed in the Class Name entry. For information about what is required to create a feature classification, see the AutoCAD® Map 3D documentation.

You can also use the Quick Select feature to select objects by their Class names and Class properties.

Quick Reference

Commands

EXPORTTOAUTOCAD

Creates a new DWG file with all AEC objects exploded.

System Variables

PROXYGRAPHICS

Specifies whether images of proxy objects are saved in the drawing.

PROXYNOTICE

Displays a notice when a proxy is created.

PROXYSHOW

Controls the display of proxy objects in a drawing.

PROXYWEBSEARCH

Specifies how the program checks for object enablers.

Extract Data from Drawings and Spreadsheets

38

You can extract property data from objects in drawings and export the output to a table or to an external file.

Overview of Data Extraction

You can extract information from objects, such as block attributes and drawing properties, to create a schedule, a parts list, or a bill of materials.

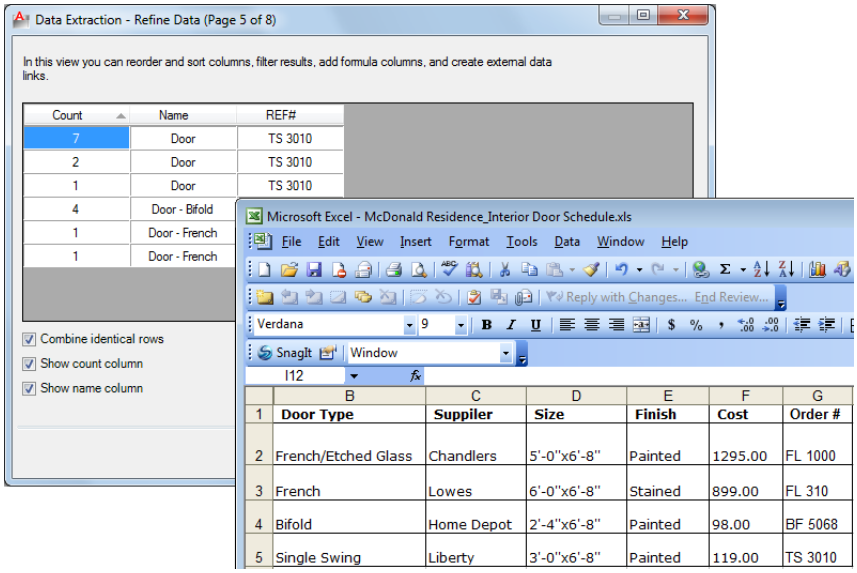
For example, let's say you have a residential construction project and you need to create a door installation schedule to place in the drawing. By using the Data Extraction wizard, you can select only those objects that are doors (blocks in this case), and extract the attribute data. Because the Data Extraction wizard also extracts property information from objects, you have more flexibility.

After you select the properties of the doors, the extracted data is displayed in the wizard in columns. You can organize and refine the information in these columns. Once the data is organized, you can format it by specifying a table style or using an existing table as a template.

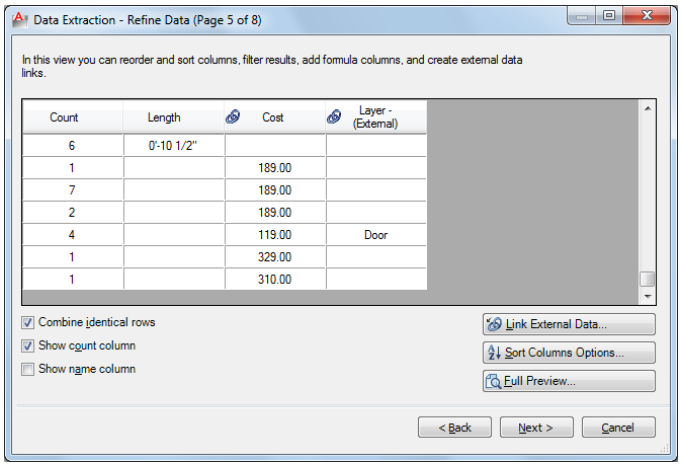
When you're satisfied with how the table looks, the extraction is complete when you insert the table into the drawing. If you need to share the extracted information with others on the project, you can output the same extracted data to an external file.

Information stored in an Excel spreadsheet can be linked and combined with the extracted *drawing data* (object property data). Expanding on the same example, you have some additional information about the doors in the your drawing in an Excel spreadsheet (supplier and cost). You select these columns

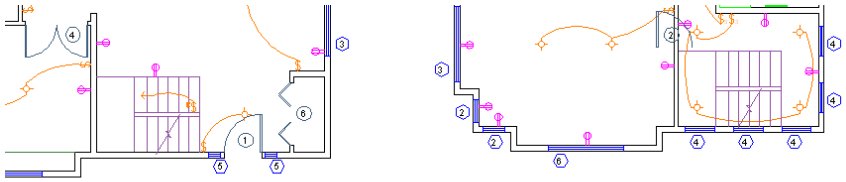
for supplier and cost in the spreadsheet and combine them with the other data in the data extraction.



The columns in the Excel spreadsheet will be added to the columns on the Refine Data page in the Data Extraction wizard.



All the columns except for Cost were added to the extracted data columns. Excel columns are designated with a symbol in the column label.



Interior Door Installation Schedule					
Count	Door Type	REF#	Size	Supplier	Finish
1	French/Etched Glass	FL 1000	5'-0"x6'-8"	Chandlers	Painted
1	French	FL 301	6'-0"x6'-8"	Lowes	Stained
4	Bifold	BF 5068	2'-4"x6'-8"	Home Depot	Painted
10	Single Swing	TS 3010	3'-0"x6'-8"	Liberty	Painted

The table inserted in the drawing contains columns from both the extracted data and from the Excel spreadsheet.

When a drawing from which property data was extracted has changed, you can be notified that the data extraction table is no longer current. For example, if some doors in the drawing were resized or removed, you can choose to be notified that the data extraction table needs to be updated. Notification is especially useful during critical times when it's important to know if the data is not current in the table, such as when plotting or publishing.

Extract Data Using the Data Extraction Wizard

You use the Data Extraction wizard to choose the data source (drawings) to extract property data from selected objects. You can output the data to a table or external file.

The Data Extraction wizard guides you through the process of

- Selecting the data source, which can be a drawing, set of drawings, or folders.
- Filtering objects (non-blocks and blocks) are extracted and selecting properties from which data is extracted.
- Organizing and refining the extracted data.
- Merging information from an Excel spreadsheet with the extracted data.
- Choosing an output format for the data (a table and/or an external file).
- Formatting the extracted data in a table by specifying a table style or selecting an existing pre-formatted table.

The first time you extract data, you are prompted to save the data extraction settings in a data extraction (DXE) file. Later, if you need to edit the data extraction, you select the DXE file, which contains all the settings (data source, selected objects and their properties, output format and table style) that you used to create the extraction. For example, if you wanted to remove some property data from the extraction, you would select the DXE file that was used to create the extraction and made the desired changes.

A data extraction file can also be used as a template to perform the same type of extraction in a different drawing. The DXE file stores drawing and folder selections, object and property selections, and formatting choices. If you need to extract the same type of information repeatedly, using a DXE file is time-saving and convenient.

You can also edit a DXE file. You can add or remove drawings, add or remove objects, or select different properties from which to extract data. Tables that reference the same DXE file, even if those tables are in other drawings, display the changes when those tables are updated.

NOTE You can use an attribute extraction (BLK) file as a template for extracting data from blocks and attributes. When using a BLK file for extracting data or editing an existing extraction, you are prompted to save the data extraction to a DXE file in order to proceed with the extraction.

See also:

- For information on identifying data extraction objects, see *Highlight External References in a Drawing* in Attach and Detach Referenced Drawings on page 1774.

To create a new data extraction table

It is recommended you save the current drawing before using the Data Extraction wizard. If the current drawing is not saved, you are prompted to save the drawing before proceeding with the data extraction.

1. Click Annotate tab ➤ Tables panel ➤ Extract Data. 
2. On 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. On 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. Enter a title for the table if necessary. Click Next.
10. On the Finish page, click Finish.
11. In the drawing, click an insertion point when prompted to create the table.

To link to a Microsoft Excel spreadsheet, see Link an Excel Spreadsheet to Access Data on page 1987.

To edit a data extraction for a table

1. Right-click a cell in the data extraction table. Click Data Extraction ➤ Edit Data Extraction Settings.

   The Define Data Source page in the Data Extraction wizard is displayed. The data extraction (DXE) file that was used for the table is automatically loaded.
2. Click Next.
3. Proceed through the pages in the wizard making the necessary changes to the data extraction.
4. On the Finish page, click Finish.

To detach a data extraction (DXE) file from a table

- Right-click a cell in the data extraction table. Click Data Extraction ➤ Detach Data Link.
  The data in the table is no longer linked to a data extraction (DXE) file, and cannot be updated.

Quick Reference

Commands

DATAEXTRACTION

Extracts drawing data and merges data from an external source to a data extraction table or external file.

System Variables

ERHIGHLIGHT

Controls whether reference names or reference objects are highlighted when their counterparts are selected in the External References palette or in the drawing window.

Start the Data Extraction

When you create a new data extraction, you are prompted to save all the specific details about the extraction to a data extraction (DXE) file. The DXE file stores drawing and folder selections, object and property selections, and formatting choices.

You can use a data extraction file as a template to perform the same type of extraction at any time. If you need to extract the same type of information repeatedly, using a DXE file is time-saving and convenient.

You can also edit an existing data extraction to add or remove drawings, add or remove objects, or select different properties from which to extract data. Tables that reference the same DXE file, even if those tables are in other drawings, will display the changes when those tables are updated.

Quick Reference

Commands

DATAEXTRACTION

Extracts drawing data and merges data from an external source to a data extraction table or external file.

Select the Source for a Data Extraction

You can extract data from individual objects in a drawing, and all objects in a single drawing, multiple drawings, or folders.

If you select a drawing, all objects in a drawing are extracted, even those that are not visible. If you have specific objects in a drawing that you want to extract data from, you can create an object selection set.

After selecting the drawing or objects for the data extraction, you can choose to filter the output by selecting options from the Data Extraction Wizard. For example, you can choose to include xrefs, or only model space objects. These options are useful for filtering the data before it is extracted. Later, you have less information to sort through.

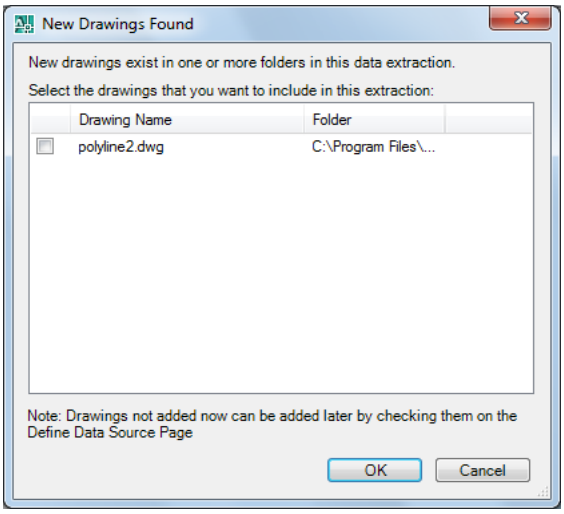
When the drawings change in a way that affects the extracted data, you can choose to be notified that the data is not current. See Update Data Extraction Table Automatically on page 1997 for more information.

NOTE The DXEVALsystem variable controls when update notification occurs.

Select Folders for Data Extraction

Using the Add Folder Options dialog box, you can select folders for the data extraction, including subfolders. If the "Automatically Include New Drawings Added In This Folder To The Data Extraction" option is not checked in the dialog box, the selected folder becomes "live", which means its contents are monitored. That means if drawings are added or to the folder, you can be notified that the data extraction is no longer current.

When new drawings are added to the folder, the New Drawings Found dialog box displays a list of the drawings when the data extraction is edited. At that point, you can select the new drawings to add to the extraction.



The name of a drawing that was added to the folder after the data extraction was performed is listed in the New Drawings Found dialog box.

Using wildcard characters (* and ?) as a search filter, data is extracted from only those drawings that match the search criteria. For example, if a folder contains an entire drawing set for a project, but you want only those drawings that contain piping data, use wild-card characters to find only those drawings. For example, **Pipe*.dwg*.

If a new drawing is added to the folder that matches the wild-card search criteria, it will not trigger an update notification because it was not included in the original data extraction. To include the new drawing, you need to edit the data extraction.

If the wildcard search criteria is removed after the data extraction has been performed, the drawings that match the search criteria are no longer monitored. If the wildcard search criteria is changed, only those drawings that match the new criteria are monitored accordingly.

To monitor selected folders for new drawings

1 Click Insert tab ➤ Linking & Extraction panel ➤ Extract Data.



2 In the Data Extraction wizard, on the Define Data Source page, click Add Folder.

3 In the Add Folder Options dialog box, clear Automatically Include New Drawings Added in This Folder to the Data Extraction.

4 To monitor subfolders, click Include Subfolders.

5 Click OK.

Quick Reference

Commands

DATAEXTRACTION

Extracts drawing data and merges data from an external source to a data extraction table or external file.

Select Objects for Data Extraction

You can select objects, including blocks and their attributes and drawing property information, from which to extract data.

After you have selected the drawings from which to extract data, the next step is to specify which objects and their properties to extract. This is the data that is output to a table or to an external file. By default, objects are extracted from both model space and paper space, including objects in blocks and xrefs. Using the Data Extraction - Additional Settings dialog box, you can change the default settings.

The Select Objects page in the wizard displays a list of objects. From this page, you can use filters to sort by object type (block or non-block). You can narrow the list of objects even more. For example, you can display only those blocks that have attributes.

Objects and their properties are always extracted using the world coordinate system (WCS), regardless if a user coordinate system (UCS) is current when the data extraction is performed.

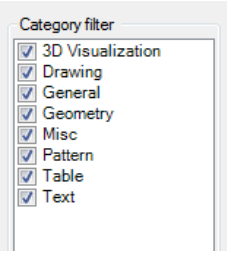
Select Object Properties to Be Extracted

After you have selected the objects, the next step is to specify which properties to extract.

The Select Properties page in the Data Extraction wizard displays a list of properties in columns from the selected objects from the Select Objects page. This is where you make a final selection of the properties from which data will be extracted and output to a columnar format. Using predefined category filters such as general, geometry, or attribute, you can narrow the property list.

Property Categories for Filtering a Data Extraction

Extracted properties are categorized for easier filtering. For example, if you wanted to see only those properties pertaining to geometry, select the Geometry filter on the Select Properties page.



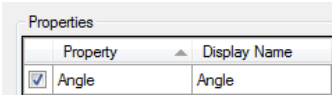
The following is a list of the filter categories for the extracted properties displayed on the Select Properties page. The categories include

- **3D Visualization**. Materials.
- **Attribute**. Attribute tags.
- **Drawing**. File name, location, size, when created and modified.
- **Dynamic Block**. Dynamic block properties.
- **General**. Color, hyperlink, layer, linetype, and so on.
- **Geometry**. Area, start point, end point, radius, and so on.
- **Misc**. Dimension style and block name.
- **Pattern**. Hatch pattern name, angle, scale, spacing, and so on.
- **Table**. Table style.
- **Text**. Single line text, multiline text, dimension text, and so on.

NOTE The category filters on the Select Properties page are the same categories that display in the Properties palette.

Property Names

After the property data is extracted, the default property names are used as the column labels in the data grid (example: Width, Length, Area, and so on). You can change these property names so that column labels are replaced with a name you specify. For example, if you extracted the length property of lines, but the column label needs to indicate the data refers to distance, you can change *Length* to *Distance* on the Display Name column on the Select Properties page.



The display name for the length property for polylines is renamed Distance.

Property Data and Unit Settings

If you select multiple drawings as the data source, and the unit setting is not the same in all the drawings, a message displays that there is a mismatch of drawing units. In this situation, the current drawing's unit settings are used for converting the numerical value of properties.

For example, if the current drawing's unit setting is set to *inches*, and another drawing's unit setting is set to *millimeters*, the extracted numerical data is automatically converted to inches. This happens even if the current drawing is not included in the data source.

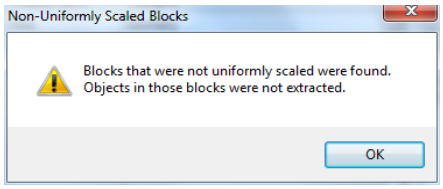
Property Types

Each property value is treated as either text or numerical data. For example, properties such as radius or length, are treated as numerical data. Attribute numerical values are treated as text. This is important to consider when using formula columns to perform arithmetic functions on extracted data.

Non-Uniformly Scaled Blocks

Objects in blocks that were not scaled uniformly cannot be extracted because the numerical values may not be correct or useful. For example, if a circle is in a block that was inserted with a non-uniform scale factor, the object is no longer a true circle.

A dialog box will display during the extraction that indicates blocks were found that were not uniformly scaled, and the objects within those blocks are not included in the extraction.



Annotative Objects

The property data for annotative objects is extracted using the current annotation scale setting.

Quick Reference

Commands

DATAEXTRACTION

Extracts drawing data and merges data from an external source to a data extraction table or external file.

Organize and Refine the Extracted Data

Extracted data can be organized, refined, and formatted before it is output to a table or external file.

The next step in the data extraction process is to refine how the data appears in the data extraction table. Once the data has been extracted from the drawing, it is displayed in columns on the Refine Data page where you can sort and reorder columns, add a formula column, and so on. You can also link to an Excel spreadsheet to add information from a spreadsheet to the extracted data.

From this page in the wizard, you can do the following organizational and refining tasks:

- Reorder, rename, hide, and sort data in columns
- Format data in cells (for example: specify a data format or convert text to lowercase)
- Insert a formula column or insert a totals footer for individual columns
- Filter data in columns
- Display or hide the Count or Name columns

- Group identical rows or display them individually
- Preview the table in a separate window
- Create a data link to an Excel spreadsheet

By right-clicking on a column label in the grid view on the Refine Data page, you access options for organizing the information the way you want it to appear in a table or in the order you want it in an external file.

Data can be displayed in either an expanded or collapsed format. In an expanded format, each property is displayed in an individual row in the data grid. In a collapsed format, duplicate property names are grouped together. A Count column is automatically created where the total number of properties are shown for each property row.

All columns that are visible in the data grid are output to the table or external file. Any columns that have been hidden are not included.

NOTE Columns that have been extracted from an Excel spreadsheet can also be hidden. These hidden columns will not be included in the data extraction table.

You can sort a single column or multiple columns at the same time using the Sort Columns dialog box.

You can insert formula columns into the data grid. Basic arithmetic operations (+, -, *, /) are available including functions from QUICKCALC and CAL commands, such as SIN, COS, and LOG. An icon displays in column labels for formula columns.



When identical rows containing numerical data are combined, where all rows containing the same property are collapsed into one row, the sum of the data is displayed.

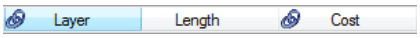
After the data has been organized, the next step is to choose an output format on the Choose Output page.

Columns from Excel Spreadsheet

If you want to include columns from an Excel spreadsheet with the extracted data, use the Link External Data dialog box, where you can set up a link to an XLS file.

If columns from an Excel spreadsheet are extracted, you can organize, manipulate, and hide the columns as data extraction columns. An icon displays

in each Excel column label so that you can easily distinguish the columns that are from a spreadsheet from those columns that are from the extracted data. The icon is not displayed in the data extraction table.



If column names in the spreadsheet contain the following characters, they are removed before the columns are displayed in the data grid on the Refine Data page:

< > / " : ; ? * | = '

When a column from an Excel spreadsheet is labeled the same as a column from the extracted drawing data, both columns are displayed; however, the Excel column label has *-(External)* added to the label name.

See also:

- Link an Excel Spreadsheet to Access Data on page 1987
- Use Formulas in Table Cells on page 1518

To view the path and file name of the data extraction file linked to a table

1. Click View tab ➤ Palettes panel ➤ Properties. 

   Alternatively, you can right-click the data extraction table and then, click Properties.

2. In the Properties palette, click the button next to Cell Data Link.

The path and name of the data extraction (DXE) file is displayed.

To rename a column label

1. Click Insert tab ➤ Linking & Extraction panel ➤ Extract Data.

2. In the Data Extraction wizard, on the Refine Data page, right-click a column label. Click Rename Column.

3. Click Rename Column.

4 In the column label field, enter a new name. Press ENTER.

To sort extracted data in a column

1 Click Insert tab ➤ Linking & Extraction panel ➤ Extract Data.



2 In the Data Extraction wizard on the Refine Data page, click Sort Columns Options.

3 In the Sort Columns dialog box, select a column name from the drop-down list.

4 In the Order column, click Ascending or Descending. Click OK.

The order of the data in the column is changed.

5 (Optional) To sort additional columns, click Add.

6 Repeat steps 2 and 3.

7 Click Insert tab ➤ Linking & Extraction panel ➤ Extract Data.



To filter extracted data in columns

1 Click Insert tab ➤ Linking & Extraction panel ➤ Extract Data.



2 In the Data Extraction wizard on the Refine Data page, right-click a column label. Click Filter Options.

3 In the Filter dialog box, select a predefined filter from the drop-down list.

4 From the drop-down list on the left, select a value.

5 In the Filter Across These Values field, clear any values so that they are not included.

To specify formatting for extracted data in a column

1. Click Insert tab ➤ Linking & Extraction panel ➤ Extract Data.



2. In the Data Extraction wizard on the Refine Data page, right-click the column label for the column that will have its data type changed. From the shortcut menu, click Set Column Data Format.
3. In the Set Cell Format dialog box, select the data type from the Data Type list (Angle, Currency, Data, Decimal Number, General, Percentage, Point, Text, or Whole Number.)
4. Depending on the selected data type, specify additional options from the right side of the dialog box.
5. Click OK.

To insert a formula column in the extracted data

1. Click Insert tab ➤ Linking & Extraction panel ➤ Extract Data.



2. In the Data Extraction wizard on the Refine Data page, right-click a column label. Click Insert Formula Column.
3. In the Insert Formula Column dialog box, in the Column Name field, enter a name for the formula column.
4. In the Columns list, double-click a column name that contains numerical data. The column name displays in the Formula field.
5. Choose an operator button [+ - * /]. The operator displays in the Formula field.
6. Add another column from the Column Name list, or enter a numerical value.
7. Click Validate.
8. Click OK.

To edit a formula in a formula column added to extracted data

1. Click Insert tab ➤ Linking & Extraction panel ➤ Extract Data.



2. In the Data Extraction wizard on the Refine Data page, right-click the column label of the formula column. Click Edit Formula Column.
3. In the Edit Formula Column dialog box, make changes to the formula.
4. Click Validate.
5. Click OK.

To insert a totals footer row for a column in a data extraction table

1. Click Insert tab ➤ Linking & Extraction panel ➤ Extract Data.



2. In the Data Extraction wizard on the Refine Data page, right-click the label for the column that will have a footer row. Click Insert Totals Footer.
3. From the flyout menu, click Sum, Max, Min, or Average.

The totals footer row is placed as the last row for the selected column.

To display the sum of numerical data in identical property rows

1. Click Insert tab ➤ Linking & Extraction panel ➤ Extract Data.



2. In the Data Extraction wizard on the Refine Data page, click Combine Identical Rows.
3. Right-click the column label. Click Combine Record Mode ➤ Sum Values.

   The sum of the numerical data is displayed.

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 data extraction tables are compared against the data source, and if the data is not current, displays an update notification.

Use Table Styles for Data Extraction Tables

A table style or existing table can be used to format a data extraction table.

If you chose the option to create a data extraction table on the Choose Output page, the Table Style page is displayed in the wizard.

On the Table Style page in the wizard, you have three options for formatting the table. You can

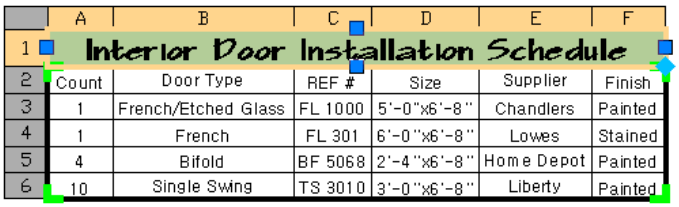
- Create a new table style.
- Select an existing table style.
- Select an existing table that is stored in a table style.

When you use create a new table style, the Table Style Dialog Box opens where you can choose formatting settings.

If there are table styles defined in the drawing, you can select an existing table style by name. With either choice, you can add a title.

If a table is already defined in a table style, its title and header cells already contain text, and the table is formatted. Using an existing table eliminates repetitive formatting and standardizes how tables are created.

Regardless of whether you use a table style or a table within a table style, the extracted data is inserted between the title, header, and footer rows (if any exist in the table).



	A	B	C	D	E	F
1	Interior Door Installation Schedule					
2	Count	Door Type	REF #	Size	Supplier	Finish
3	1	French/Etched Glass	FL 1000	5'-0"x6'-8"	Chandlers	Painted
4	1	French	FL 301	6'-0"x6'-8"	Lowes	Stained
5	4	Bifold	BF 5068	2'-4"x6'-8"	Home Depot	Painted
6	10	Single Swing	TS 3010	3'-0"x6'-8"	Liberty	Painted

The bold line around the cells indicate the boundary of the cells that contain the extracted data.

Before completing the extraction, you can see the results of your formatting choices in a separate preview window.

See also:

- Work with Table Styles on page 1511

Quick Reference

Commands

DATAEXTRACTION

Extracts drawing data and merges data from an external source to a data extraction table or external file.

Link an Excel Spreadsheet to Access Data

Information in a Microsoft Excel spreadsheet can be combined with extracted data from drawings.

By linking to an Excel spreadsheet, you can include information from an entire worksheet, or part of a worksheet, referred to as a *named range*. Using the Link External Data dialog box, you can

- Specify a data link by selecting an XLS file to which the extracted data from the drawing will be linked.
- Match the extracted drawing data and the spreadsheet data. (See Understand Data Linking and Matching for more information.)
- Select the columns in the XLS file to merge with the extracted drawing data.

On the Refine Data page, you access the Link External Data dialog box; you can set up a link to an entire Excel spreadsheet, or to a range of rows and columns, to the extracted data.

Before you can link a spreadsheet, Microsoft Excel needs to be installed on the same computer on which the data extraction is performed. If Microsoft Excel is not installed, or if the XLS file is not saved locally on the computer that is performing the data extraction, a data link cannot be established.

If a linked spreadsheet has been changed, such as a row or column has been added, the table in your drawing can be updated accordingly using the DATALINKUPDATE command. Likewise, if a change is made to the table in the drawing, the linked spreadsheet can be updated using the same command.

NOTE To delete a data link, use the DATALINK command. You cannot delete a data link from the External References Palette.

Understand Data Linking and Matching

Data linking is the process of selecting a Microsoft Excel spreadsheet with the purpose of merging all or some of its contents with extracted drawing data. See Link a Table to External Data on page 1507 for more information about data linking.

Data matching creates a relationship between two sets of data; the data in a column from the extracted drawing data and the data in a column from an Excel spreadsheet.

For example, you have a simple bill of materials (BOM). Three columns of data are extracted from a drawing. These columns contain data for: *Part #*, *Revision*, and *Quantity*. You want to include two additional columns of data in the BOM from a spreadsheet. These columns contain data for *Cost* and *Supplier*.

In this example, the spreadsheet contains a column of part numbers labeled *Catalog Number* that matches the *Part Number* column in the extracted data. In the Link External Data dialog box, you select *Part #* as the column for the drawing data and *Catalog Number* as the column for the external data. Because there is at least one cell of matching data in the drawing data column and the external data column, the data extraction feature "knows" how both sources are tied together.

You can test if the data matching is valid by clicking the Check Match button from the Link External Data dialog box. If the match is unsuccessful, a warning message indicates what the problem is. In some situations, the drawing data might not link correctly to the data in the spreadsheet. See Data Matching Validation for more information.

Validate Matched Data

The data matching validation process fails when it cannot find a valid match between the drawing data and external data (spreadsheet). The following error messages will display:

No match was found between any values in the drawing data and external data.

- If the problem is not resolved, the external data will not be merged. Compare the spreadsheet column data to the drawing data column for any matching values. For example, the numerical data in the drawing data may be set to a different precision level than in the external data.

The data column in the external data does not contain unique values.

- The data in the external data column contains one or more cells that contain identical data. This condition makes it so that one match cannot be configured between the drawing data and the external data.

See also:

- Link a Table to External Data on page 1507
- Overview of Data Extraction on page 1969

To link data to an Excel spreadsheet in a data extraction table

1. In the Data Extraction wizard on the Refine Data page, click Link External Data.
2. In the Link External Data dialog box, click Launch Data Link Manager.
3. In the Data Link Manager tree view, select Click the Create a New Excel Data Link option.
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 a link option (whole sheet, range, or Excel named range). Click OK.
7. Select the new data link in the Data Link Manager tree view. Click OK.

8. In the Link External Data dialog box, in the Drawing Data Column list, specify the column you want to link.
9. In the External Data Column list, specify the Excel column you want to link.
10. Click Check Match. If an error displays, see Data Matching Validation.
11. From the Select External Data Columns to Include, select the Excel columns from the list you want to add to the extracted data.
12. Click OK.

To set up a named range in Microsoft Excel to use in a data extraction

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. Enter a name for the range of cells in the Name Box, then press ENTER.
4. Repeat steps 2 and 3, if desired, to specify additional linked ranges.
5. On the File menu (Microsoft Excel), click Save.

To view the data link from a data extraction table to an Excel spreadsheet

- Right-click the data extraction table. Click Properties.

The Cell Data Link field in the Properties palette displays the name of the data link associated with the table.

To detach a link from a data extraction table to an Excel spreadsheet

- Select the data extraction table and right-click. Click Data Links ➤ Detach Data Link.

The data link to the Excel spreadsheet is removed from the data extraction table.

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 data extraction tables are compared against the data source, and if the data is not current, displays an update notification.

Output Extracted Data

You can export extracted data to a table or an external file.

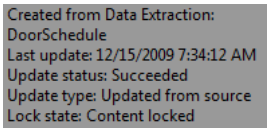
On the Choose Output page in the Data Extraction wizard, you can output the extracted data to a data extraction table, to an external file, or both.

Output Extracted Data to a Table

By choosing the Insert Data Extraction Table into Drawing option on the Choose Output page, you are prompted to insert the table into the current drawing when the extraction is complete. The table is created using the table style you selected, and is populated with the data that was extracted from the data source. If an Excel spreadsheet is linked, the selected columns from the spreadsheet are included in the table.

Although a data extraction table looks like a table that was created using the TABLE command, there are some key differences.

- A tooltip displays information about the data extraction when the table is selected.
  When a drawing containing a data extraction table is opened in AutoCAD LT, the tooltip text indicates the table contains extracted data. You cannot update the extracted data or change a data link, if one exists



- The cells are locked and their contents cannot be modified unless they are unlocked. (Be default, formatting for cells is unlocked.)

Cells in a data extraction table are locked by default to prevent editing. Individual cells or the entire table can be unlocked if you want to make data changes. See Modify a Data Extraction Table on page 1999 for more information.

See also:

- Modify a Data Extraction Table on page 1999

Output Extracted Data to a File

The same data that is extracted to a table can be exported to an external file. On the Choose Output page, select the Output Data to an External File option. If a data link was established to an Excel spreadsheet, the data in the spreadsheet is also exported.

You can export the data to the following file formats:

- **Microsoft Excel (XLS)**.
- **Microsoft Access (MDB)**.
- **Comma-Separated (CSV)**. A *.csv* file is a data file in which the fields are separated by the comma character and records (rows) are separated by new lines. This format is suitable for input into most spreadsheet applications.
- **Tab-Separated (TXT)**. A *txt* file is a data file in which the fields are separated by tab characters. This format is suitable for input into a spreadsheet application. Formatting information is not included.

NOTE When the characters period (.), comma (,), or pound sign (#) are written to an Excel or Access file, they are replaced with their Unicode representation.

By right-clicking a data extraction table, you can access the Export option to output the contents of the entire table to a comma-separated (CSV) file. You can also use the -DATAEXTRACTION command to export data in the drawing using a BLK or DXE template file.

A maximum of 255 columns can be output when exporting to either a Microsoft Excel (XLS) file or Microsoft Access (MDB) file. If the extracted data exceeds 255 columns, a dialog box is displayed after clicking Next on the Choose Output page when either file format is chosen. The dialog box indicates only the first 255 columns of data will be exported. An unlimited number of

columns can be exported to the comma-separated (CSV) and tab-separated (TXT) file formats.

To export extracted data to an external file using the Data Extraction wizard

1. Click Annotate tab ➤ Tables panel ➤ Extract Data. 
2. In the Data Extraction wizard, click Next on each page until the Choose Output page is displayed.
3. Click Output Data to External File.
4. Click the Browse button. Select a file format from the drop-down list.
5. Click Next. Proceed through the wizard until the Finish page displays.
6. Click Finish.

The external file is created.

To export extracted data in a table to a comma-separated (CSV) file

1. Select the table and right-click. Click Export.
2. In the Export Data dialog box, enter a file name or accept the default file name. Click Save.

The external file is created as a CSV file.

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 data extraction tables are compared against the data source, and if the data is not current, displays an update notification.

Update Extracted Data

Extracted data in a table can be updated either manually or automatically.

Overview of Updating Extracted Data

Extracted data can be updated so that it is current with the data source. You can manually update the extracted data, whether it is output to a data extraction table or to an external file, or rely on a notification to display when using certain trigger commands, such as SAVE or PLOT. You can then update the table from the Data Extraction - Outdated Table dialog box.

NOTE Extracted data in external files is not checked to see if it is current with the data source, and notification does not occur.

Update an Attribute Extraction Table

You can update the contents of a table that was created using the Attribute Extraction wizard. Before it can be updated, the table first needs to be converted to a data extraction table.

NOTE The width and height of columns of the attribute extraction table are not retained after it is converted to a data extraction table.

See also:

- Select the Source for a Data Extraction on page 1975

To edit the data extraction settings for a data extraction table

1. In the drawing, right-click the data extraction table. Click Data Extraction ➤ Edit Data Extraction Settings.
2. In the Data Extraction wizard, on the Define Data source page, make any changes.
3. Click Next. Proceed through the wizard and make the necessary changes until the Finish page is displayed.
4. Click Finish.

To convert an attribute extraction table to a data extraction table

1. Right-click the attribute extraction table. Click Convert to Data Extraction Table.
2. In the dialog box, enter a name for the data extraction (DXE) file and click Save.

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 data extraction tables are compared against the data source, and if the data is not current, displays an update notification.

Update Extracted Data Manually

You can update the extracted data in a table or external file manually.

If the drawings that were used for the data extraction have changed, you can manually update the data. To update a data extraction table, you can use either of two methods.

- Open the drawing that contains the table and use the Data Extraction wizard to edit the DXE file that contains the extraction settings.
- Right-click the data extraction table to access its shortcut menu. Click Data Extraction ➤ Update Data Extraction.

To update the extracted data that was output to an external file, you use the Data Extraction wizard and specify the DXE file that was used for the extraction.

NOTE The *.dxe* file is saved with the file name and location that was specified when a new data extraction was created.

Using the data extraction (DXE) file, you can also edit an existing data extraction to add or remove drawings, add or remove objects, or select different properties from which to extract data. Tables that reference the same DXE file, even if those tables are in other drawings, will display the changes when those tables are updated.

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 Preference tab.

To modify an existing data extraction in an external file

1. In the drawing, right-click the data extraction table. Click Data Extraction ➤ Edit Data Extraction Settings.
2. In the Data Extraction wizard, on the Define Data Source page, make any changes.
3. Click Next. Proceed through the wizard and make the necessary changes until the Finish page displays.
4. Click Finish.

To change a data link for a data extraction table

1. In the drawing, right-click the data extraction table. Click Data Extraction ➤ Edit Data Extraction Settings.
2. In the Data Extraction wizard, on the Define Data Source page, click Next and proceed to the Refine Data page displays.
3. Click Link External Data.
4. Click Launch Data Link Manager.
5. Click the Create a New Excel Data Link option.
6. In the Enter Data Link Name dialog box, enter a name for the data link. Click OK.
7. Click the [...] button to browse for the *.xls* file.
8. In the New Excel Data Link dialog box, select a link option (whole sheet, range, or Excel named range). Click OK.
9. Select the new data link in the Data Link Manager tree view. Click OK.

To view a data extraction (DXE) file associated with a data extraction table

- In the drawing, right-click a cell in the data extraction table. Click Cell Data Link.
  The path and the DXE file name is displayed.

To view a data extraction (DXE) file and linked Excel (XLS) file associated with a drawing

- Click Insert ➤ External References.

The file name and path are displayed. If a data link exists to a Microsoft Excel (XLS) file, information about the data link is displayed on the Details pane.

To detach a data link for a data extraction table

- In the drawing, right-click the data extraction table. Click Data Extraction ➤ Detach Data Link.
  The .DXE file is detached from the data extraction table.

To manually update a data extraction table

- Right-click the data extraction table. Click Data Extraction ➤ Update Data Extraction.

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 data extraction tables are compared against the data source, and if the data is not current, displays an update notification.

Update Data Extraction Table Automatically

You can specify to be notified when extracted data has changed.

When information in the data source has changed that affects the extracted data in a table, you are notified the data is not current. (Notification does not occur with attribute extraction tables.) Depending on how the DXEVAL system variable is set, the Data Extraction - Outdated Table dialog box displays when certain commands are used, such as OPEN or SAVE. By default, DXEVAL is set to check if the extracted data is not current without automatically updating it when the PLOTor ETRANSMIT commands are used.

IMPORTANT When DXEVAL = 8 (update on publish) and BACKGROUNDPLOT is set to 2 or 3, then you will be prompted to update when publishing the files. However, the tables in the DWF file will not be updated. To include table updates in the published files, set BACKGROUNDPLOT to 0 or 1.

You can set auto-updating to occur with these commands: SAVE, PLOT, PUBLISH, or ETRANSMIT. When auto-updating is enabled, the extracted data is automatically checked to see if it is current with the data source. If not, the update process occurs without user intervention. With PLOT or ETRANSMIT, the data extraction is updated prior to the plotting or electronically transmitting the drawings, which are not saved.

When you open multiple drawings at the same time, and the DXEVAL setting in each drawing is 1 (to check for updates as the drawing is open), a dialog box displays for each drawing that contains a data extraction table that is not current. You can choose to update the drawing or to skip all updates.

You can run the update later from the shortcut menu for the data extraction table.

If the cells in a data extraction table are edited in a previous release of the program, and then opened in the current version, the table cells that were edited are unlocked. When the table is updated in the current release, a dialog box displays indicating some cells were overwritten. You can choose to overwrite the cells or skip the cells during the update.

NOTE It is recommended you turn on notification before doing a final save, plot, publish, electronic transmittal, or archive.

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 data extraction tables are compared against the data source, and if the data is not current, displays an update notification.

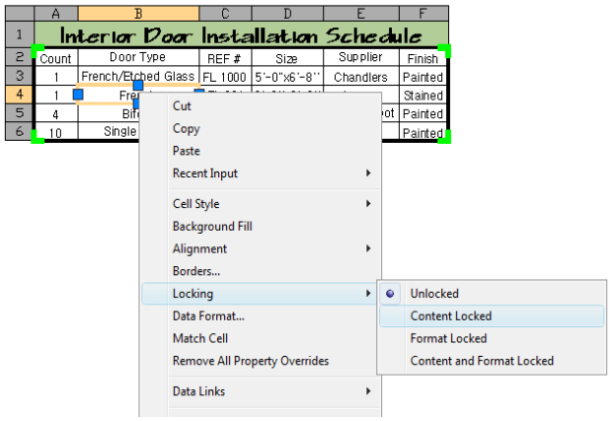
Modify a Data Extraction Table

You can modify a data extraction table by changing the formatting, adding columns and rows, or editing data in the cells.

Once a data extraction table is inserted into the drawing, you can make formatting changes, add columns and rows, and edit cells containing extracted data.

Any formatting, structural, or data changes that are made are retained after the table is updated. For example, if you added shading to a header row, or the formatting of text in certain columns changed, those changes are not lost when the table is updated.

By default, all cells in a data extraction table are locked for editing and unlocked for formatting changes. You can unlock or lock cells for data and formatting edits by accessing the table's shortcut menu.



Add Columns and Rows

You can add columns and rows to a data extraction table without affecting the extracted data. When you select a table, the lines around cells that contain extracted data is bold. Columns and rows can only be added outside this perimeter. You should be aware that blank cells may be created as a result if

additional data is extracted during an update, or conversely, there are fewer columns or rows containing extracted data.

For example, if you added a row and column to a table after you inserted into the drawing, the area populated with extracted data will be expanded by an additional column after the table is updated. The column and row that you manually added are still part of the table, but a blank cell is created where a void exists in the table.

Using the same example, if the table was updated and the number of columns and rows containing extracted data were reduced, the columns and rows you inserted would remain in the same place. The area that originally contained data extraction columns and rows is now replaced with blank cells.

Edit Cell Content

Data cells are locked by default in order to prevent unintentional changes. You can manually unlock a cell to edit its contents. When the table updates, you are given a choice to overwrite the edited cells or choose to have the update not update the cells.

See also:

- Create and Modify Tables on page 1503

To edit/unlock the cells in a data extraction table

- In the drawing, right-click a cell in the data extraction table. Click Cell Locking ➤ Data.

The cells are unlocked for editing.

To edit/unlock the cells in an entire data extraction table

- In the drawing, right-click a cell in the data extraction table. Click Cell Locking ➤ All ➤ Unlocked.

All the data cells in the table are unlocked for editing.

To edit the text in an unlocked cell in a table

- In the drawing, right-click the cell in the data extraction table. Click Edit Cell Text.

All the data cells in the table are unlocked for editing.

To identify which cells in a table are unlocked or locked

1. In the drawing, right-click a cell in the data extraction table. Click Properties.
2. In the Properties palette, look at the Cell Locking field.

One of the following states display

- If the cell is unlocked for data and formatting, *Unlocked* displays.
- If the cell is locked for data, *Data Locked* displays.
- If the cell is locked for formatting, *Format Locked* displays.
- If the cell is locked for data and formatting, *Data and Format Locked* displays.

To add a column to a data extraction table

1. Right-click in a table cell in the first column or last column.
2. To insert a column before the first column, click Columns ➤ Insert Left. To insert a column after the last column, click Columns ➤ Insert Right.

To add a row to a data extraction table

1. Right-click in a table cell in the first row or last row.
2. To insert a row above the first row, click Rows ➤ Insert above. To insert a column after the last row, click Rows ➤ Insert Below.

To equally size all columns and rows in a table

- Select the table and right-click. Click Size Columns Equally or Size Rows Equally.

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 data extraction tables are compared against the data source, and if the data is not current, displays an update notification.

Detach a Data Extraction Table

To completely remove a data extraction table, you need to detach it.

You can delete a data extraction table from the External References Palette by detaching it.

Quick Reference

Commands

DATAEXTRACTION

Extracts drawing data and merges data from an external source to a data extraction table or external file.

EXTERNALREFERENCES

Opens the External References palette.

Access External Databases 39

You can use the program to associate, or link, data stored in external database programs such as dBASE, Oracle, and Microsoft Access with graphical objects. You can use all of the external database features without prior knowledge of databases or query languages. This section, however, does not cover in detail the theory of database design or the semantics of Structured Query Language (SQL).

Overview of Using External Databases

When you link database records to graphical objects in a drawing, you can use powerful database queries to filter and sort information. You can present the results of a query graphically.

A database is a collection of logically related information, usually presented in a table format similar to a typical spreadsheet. The table below is an example of a simple database that might be used by a Facilities department to schedule maintenance work. It consists of four columns, or *fields*, labeled Maintenance task, Estimated hours, Frequency, and Assigned to. Individual fields, or *cells*, store data for a particular variable. Individual rows within the database table are called *records*.

Maintenance database table

Maintenance task	Estimated hours	Frequency	Assigned to
ELEVATOR SRV	3.5	QUARTERLY	RON SMITH
CARPET VAC	5.25	WEEKLY	VICKI ROBERTS
A/C CHECK	2.00	BIWEEKLY	AMY REED
AIR FILTER	3.00	MONTHLY	DAT TAN

Most database systems are extremely flexible and can be easily modified. You can add new fields or remove existing ones from database tables. Likewise, you can add or delete records from database tables and edit existing records. With many databases, you can also define *relationships* between multiple database tables. For example, suppose you have two database tables that share a common field, Room ID. Using a relational database, you can join these two database tables using their common field. You don't have to maintain and input duplicate data across multiple database tables.

You can use the program to associate data contained in an external database table with graphical objects through the process of linking. *Links* are pointers to a database table that reference data from one or more records in that table. You can also use the program to attach *labels* that display data from selected database table fields as text objects in your drawing.

The database connectivity feature provides the following:

- An external configuration utility that enables the program to access the data contained in a particular database system
- A dbConnect Manager where you can associate links, labels, and queries with drawings
- A Data View window that displays records from a database table within an program session
- A Query Editor where you can construct, execute, and store SQL queries
- A migration tool that converts links and displayable attributes to AutoCAD 2000 or later format from files created in earlier releases
- A Link Select operation that creates iterative selection sets based on queries and graphical objects

Quick Reference

Commands

DBCONNECT

Provides an interface to external database tables.

System Variables

DBCSTATE

Indicates whether the dbConnect Manager is open or closed.

Access a Database from Within Drawings

When you have configured a data source, you can view and edit its records from within a drawing file.

Configure a Database for Use with Drawing Files

The configuration process involves creating a new *data source* that points to a collection of data and provides information about the drivers necessary to access it.

Before you can access an external database from within the program, you must configure it using the Microsoft ODBC (Open Database Connectivity) and OLE DB programs. Using ODBC and OLE DB, the program can utilize data from other applications, regardless of the format that it's stored in or the database platform on which it is created.

A data source may be either an individual database table or a collection of them stored in an *environment, catalog,* or *schema*. Environments, catalogs, and schemas are hierarchical database elements used by most database management systems that help you organize your data. They are in many ways analogous to a Windows-based directory structure: you can think of the environment as a folder that holds additional subdirectories of catalogs. Catalogs in turn contain additional subdirectories of schemas. Each schema is a collection of database tables.

The configuration process varies slightly for different database systems. For example, server-based databases such as Oracle and Microsoft SQL Server require that you enter a valid user name and password and specify the network location where the database is located. File-based systems such as Microsoft Access and dBASE III do not require this information. Because of these variations, it is not possible to provide a generic configuration procedure that works for all databases. Individual procedures for configuring all the databases that the program supports are provided in the Configure External Databases. You should also refer to the documentation for your particular database system and the Microsoft Help files for ODBC and OLE DB for additional information.

The connectivity feature supports the following external applications:

- Microsoft Access
- dBASE
- Microsoft Excel
- Oracle

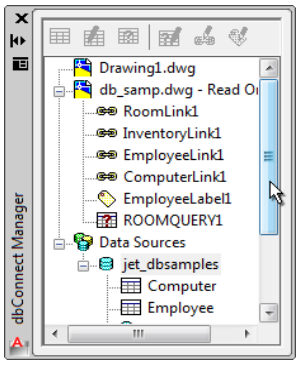
- Paradox
- Microsoft Visual FoxPro
- SQL Server

NOTE The 64-bit release of AutoCAD does not support the use of Microsoft Jet 4.0 OLE DB Provider (for *.MDB* connectivity) and Microsoft OLE DB Provider for ODBC Drivers (for *.XLS* connectivity). For more information, see Substituting SQL Server for OLE DB.

The program includes a Microsoft Access sample database file and a preconfigured direct driver *(jet_dbsamples.udl)* that you can use to work with the database. You do not need to have Access installed. The file is located in the *Sample* folder, which is located in the installation directory. If you moved the *Sample* folder to a different location, you must update the path in the *jet_dbsamples.udl* configuration file before you can work with the database file.

The dbConnect Manager

The dbConnect Manager is a dockable, resizable window that contains a set of buttons and a tree view. In the dbConnect Manager, you can open the Data View window to view or edit a database table. You can also associate various *database objects* (such as link templates, label templates, and queries) with a drawing.



The tree view of the dbConnect Manager contains the following nodes:

- **Drawing Nodes.** Display each open drawing. Each drawing's node shows all database objects that are associated with the drawing.
- **Data Sources Node.** Displays all configured data sources on your system.

All of the various nodes and database objects in the dbConnect Manager have shortcut menu options associated with them. For example, you can use shortcut menu options to configure a data source or edit a stored query.

To open and close the dbConnect Manager

- Click Tools menu ➤ Palettes ➤ DbConnect.

To update the jet_dbsamples.udl configuration file with new information

1. Click Tools menu ➤ Palettes ➤ DbConnect.
2. In the dbConnect Manager, in the tree view, right-click Data Sources. Click Configure Data Source.
3. In the Configure a Data Source dialog box, select Jet_dbsamples from the Data Sources list. Click OK.
4. In the Data Link Properties dialog box, Connection tab, click the [...] button.
5. In the Select Access Database dialog box, navigate to and select the *db_samples.mdb* file. Click Open.

   This file is located in the *Sample* folder.
6. Click Test Connection to verify that your connection is working properly.
7. If the connection fails, verify that you selected the appropriate file.
8. Click OK to close each dialog box.

Quick Reference

Commands

DBCONNECT

Provides an interface to external database tables.

System Variables

DBCSTATE

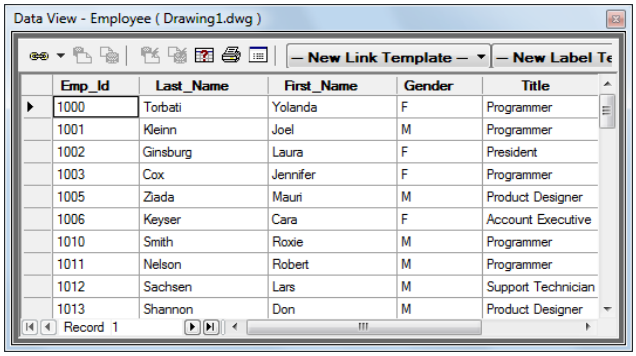
Indicates whether the dbConnect Manager is open or closed.

View Data in a Database Table

When you have configured a data source, you can use the Data View window to view database records from within the program.

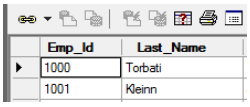
Once you have configured a data source, you can access its database tables from within the program. The Data View is a window where you can view and edit database records. You can open database tables in Read-only mode or in Edit mode. You cannot add, delete, or edit records in a database table that you open in Read-only mode. Opening a database table establishes a *connection* to its parent database. Some database systems require that you enter a valid user name and password before you can connect to them.

The Data View window is a spreadsheet-like environment that presents a database table's records. With the scroll bars and navigation buttons at the right and at the bottom of the window, you can move through the record set.



The *grid* in the Data View window contains the following elements:

- **Column header.** Selects all records in a column when you click it.
- **Record header.** Selects an individual record when you click it.
- **Grid cells.** Selects one field of a given record when you click it.
- **Grid header.** Selects the entire database table.



The Data View window provides a great deal of flexibility in how records are displayed. You can

- Move or resize columns
- Hide columns to remove them from the Data View window display
- Sort column data in ascending or descending order
- Freeze a column or combination of columns so that they stay in place and do not scroll with the horizontal scroll bar
- Align text within individual columns
- Apply a variety of font styles and sizes to the Data View window display

Column formatting and database table sorts in the Data View window are not stored. This information is discarded when you open a new database table or close the current table. If you want to create and save a formatted report, you must copy the contents of the Data View window to the Clipboard and paste it into an external program such as Microsoft Excel. You can then use the external program to apply and save the desired formatting. For details about creating a template, refer to the documentation of the external program that you are using to format the data.

Database Views and External Queries

Some database management systems support *views*. Views provide additional ways to display data from a database table. For example, you can use a view to display a subset of the overall database table. You can also use views to define relationships between database tables and create a new table that contains combined records from the original database tables.

Database systems that do not support views can produce similar results with queries. Refer to your database system documentation for information about creating and working with views and queries.

Views and external queries are displayed in the Data Sources node of the dbConnect Manager. *External queries* are queries created and stored outside of the program but within an external database system. You can work with external queries and views as you would work with any other database table. You can view or edit them in the Data View window, construct queries based on them, and create links and labels based on their records.

To open a database table for viewing or editing

1 Click Toolsmenu ➤ Palettes ➤ DbConnect.

2 Do one of the following:

- To open a database table in Read-only mode, click DbConnect menu ➤ View Data ➤ View External Table.
- To open a database table in Edit mode, click DbConnect menu ➤ View Data ➤ Edit External Table.

If required by the database to which you are connecting, you are prompted to enter a valid user name and password.

3 In the Select Data Object dialog box, navigate to and select the database table you want to open. Click OK.

NOTE Some database systems require that you obtain access privileges before you can view or edit database tables. Refer to the documentation for your database system, or check with your database administrator.

To open a database table directly from the Data Sources node of the dbConnect Manager

- Double-click a database table in the dbConnect Manager.
  By default, double-clicked database tables are opened in Edit mode. You can change the default double-click behavior to Read-only mode, if desired.

To change the double-click open setting for database tables

1 Click Tools menu ➤ Options.

2 On the System tab, select Open Tables in Read-only Mode.

3 Click OK.

To move a column in a database table

1 Click the column header of the column that you want to move.

2 Click the column header again and drag it to the desired location.

To resize a column in a database table

1 In the Data View window, select the grid line at the right border of the column header of the column you want to resize.

2 Drag to the right to enlarge the column, or to the left to shrink it.

To hide a column in a database table

1. In the Data View window, select the column header of the column that you want to hide.
2. Right-click the column header. Click Hide.

To display all hidden columns in a database table

- Right-click any column header in the Data View window. Click Unhide All.

To specify a sort order for the Data View window display

1. Right-click any column header in the Data View window. Click Sort.
2. Select a column to sort in the Sort By list. Select Ascending or Descending to specify a sort order.

   Ascending order sorts the column starting with the smallest value and ending with the largest value. Descending order sorts the column starting with the largest value and ending with the smallest value.
3. If desired, select an additional column to sort in the Then By list. Select the Ascending or Descending option to specify a sort order.

   The sort operation sorts all database table records according to the value specified in the Sort By list, and then continues to sort based on the value specified in the Then By list.
4. Repeat step 2 for any additional fields that you want to add to the sort.

   You can sort by a combination of up to five table columns at a time.

To freeze a column or columns

1. In the Data View window, select one or more columns to freeze.
2. Right-click one of the selected column headers. Click Freeze.

NOTE The Freeze option can be applied only to contiguous columns. If not all of the columns you want to freeze are contiguous, move them as described in To move a column in a database table.

To unfreeze all frozen columns

- Right-click any column header in the Data View window. Click Unfreeze All.

To align text in a column or columns

1. Select one or more columns in the Data View window whose text you want to align.
2. Right-click any column header. Click Align.
3. Select one of the following to align the text:
   - **Standard.** Right-aligns numeric fields and left-aligns all other fields.
   - **Left.** Left-aligns the column cells.
   - **Center.** Center-aligns the column cells.
   - **Right.** Right-aligns the column cells.

To select a font size and style for the Data View

1. In the Data View window, right-click the empty, top-left square of the database table. Click Format.
2. Select the font settings you want to use. Click OK.

To export data from the Data View window

1. Open a database table by double-clicking it in the dbConnect Manager.
2. Select the records that you want to export from the Data View window.
3. Right-click in any cell. Click Copy.
4. Open the external application to which you want to export the data.

   Click Home tab ➤ Utilities panel ➤ Paste. 

   The text is inserted into the external application in tab-delimited format.

Quick Reference

Commands

DBCONNECT

Provides an interface to external database tables.

Edit Data in a Database Table

In the Data View window, you can search database records for a particular value, and you can add, delete, or change records.

If you have opened a database table in Edit mode, you can edit its data in a number of ways. You can add or delete records, or edit a record's values. You can also search a particular column for occurrences of a specific value.

NOTE The Find option limits its search to records contained in the same column as the currently selected cell. There is no way to issue a global Find command that searches the entire database table for a specific value. You must select individual cells in any additional columns you want to search and click Find.

Changes that you make to individual records in a database table are not written to the database until you *commit* them. The Commit option updates the database table currently loaded in the Data View window with any changes that you make during an editing session and closes the Data View window. The Restore option undoes any changes you made in the Data View window during an editing session and closes the Data View window.

NOTE If you quit the program, open a new database table, or close the Data View window before committing changes you've made to a database table, the program automatically commits all changes you've made during an editing session.

To help you track your editing during a work session, changed records are marked with an edit icon. You can easily review your changes before committing them to the source database table.

To edit a database record

1. Select a cell to edit in the Data View window.
2. Enter a new value in the cell.
3. Repeat as necessary for any additional cells that you want to edit.

   You can use the keyboard arrow keys to move from cell to cell.

To add a new database record

1. In the Data View window, right-click a record header (the empty square at the left of the row). Click Add New Record.

   A new record with blank field values is added to the end of the current record set.

2. Select a cell in the new record and enter a value.
3. Repeat step 2 for any additional fields.

To delete a database record

- In the Data View window, right-click a record header (the empty square at the left of the row) of the record that you want to delete. Click Delete Record.

To search the Data View window for a particular value

1. Right-click a cell in the column that you want to search. Click Find.
2. Enter a value to search for in Find What.
3. Select a direction to search the Data View window:
   - **Up.** Searches the column records from the selected cell through the first record.
   - **Down.** Searches the column records from the selected cell through the last record.
4. Click Find Next.
5. Repeat step 4 to find additional occurrences of the specified value.

NOTE The Find option limits its search to records contained in the same column as the currently selected cell. There is no way to issue a global Find command that searches the entire database table for a specific value. You must select individual cells in any additional columns you want to search, and then click Find.

To search the Data View window for a particular value to replace

1. Right-click a cell in the column that you want to search. Click Replace.
2. Enter a value to search for in Find What.
3. Enter a replacement value in Replace With.
4. Click Find Next.
5. Do one of the following:
   - Click Replace to replace the current occurrence of the search value.
   - Click Replace All to replace all occurrences of the search value.

6 Repeat steps 4 and 5 to replace additional occurrences of the value you're searching for.

NOTE The Replace option limits its search to records contained in the same column as the currently selected cell. There is no way to issue a global Replace command that searches the entire database table for a specific value. You must select individual cells in any additional columns you want to search, and then click Replace again.

To commit Data View window editing

- In the Data View window, right-click the empty, top-left square of the database table. Click Commit.

To restore original values to the Data View window

- In the Data View window, right-click the empty, top-left square of the database table. Click Restore.

NOTE If you quit the program, or open a new database table, or close the Data View window before committing changes you've made to a database table, the program automatically commits all changes you've made during an editing session.

Quick Reference

Commands

DBCONNECT

Provides an interface to external database tables.

Link Database Records to Graphical Objects

Links connect graphical objects with tables in an external database.

Create and Edit Links and Link Templates

Each link requires a link template, which identifies what fields from a table are associated with the links that use that template.

The primary function of the database connectivity feature is to associate external data with program graphical objects. You can, for example, associate

information contained in a room database with a polyline object representing a room boundary in a drawing. You establish the association between the graphical object and the database table by creating a *link*, which references one or more records stored in the database table. You cannot create links to nongraphical objects such as layers and linetypes.

Links are tightly connected with the graphical objects they are associated with. If you move or copy a linked object, the link is moved or copied with it. If you delete a linked object, the link is also deleted.

When you create a link, a dynamic relationship is established between the database record and the object. If, for example, you convert a room from a storage area to an office in the database table, the program provides a mechanism to update the information stored in the drawing so that it matches the database table.

To establish links between database table records and graphical objects, you must first create a *link template*. Link templates identify what fields from a database table are associated with the links that share that template. For example, you can create a link template that uses the field Room Number from a room database table. You can then use this link template to create links that point to different records in the database table.

Link templates also function as shortcuts that point to the database tables that they're based on. You can use the link templates that are associated with a drawing to open the database tables that they reference for viewing or editing. This is particularly useful if you have a large number of data sources configured on your system. Rather than scrolling through a list of data sources each time you want to locate an individual database table, you can open it directly from the drawing node of the drawing that it is associated with. To open a link template's database table, right-click a link template in the dbConnect Manager. Click View Table or Edit Table.

You can attach multiple links using different link templates to a single graphical object. This is useful when you want to associate data from multiple database tables with an individual object.

Links in External References and Blocks

Links stored in externally referenced drawings cannot be viewed or modified in the master drawing that they're attached to. This information is only available when you open the source drawing directly.

Links contained in blocks are available, but can only be accessed through the REFEDIT command.

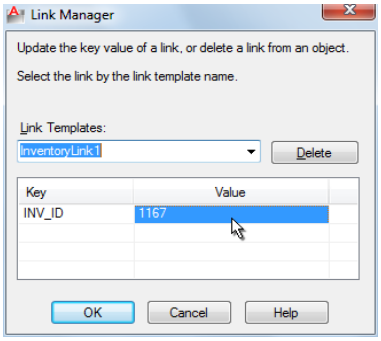
Edit Link Templates

Occasionally after creating a link template, you may want to edit it. For example, you might decide to add an additional key field or remove an existing one. You may also need to update a link template if any changes are made to the data source it points to. For example, you might rename a database table or move it to a different location.

NOTE You can only change the key fields of link templates that don't have any links defined in their associated drawing. Changing the key fields of a link template invalidates its links and breaks their association to the database table. To edit a link template with links already defined, you must first delete all links based on the link template from its associated drawing.

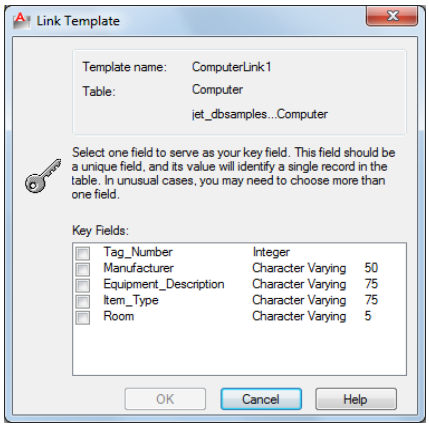
Edit Links

After you have created links, you may periodically need to update their key field values. For example, you might change the numbering scheme for a group of offices, or reallocate some rooms from the accounting department to the sales department. You can use the Link Manager to enter new key values for a selected link. The key values must reference a record that already exists in the source database table.



To create a link template in the current drawing

1. Click Toolsmenu ➤ Palettes ➤ DbConnect.
2. In the dbConnect Manager, in the tree view, right-click a data source file. Click New Link Template.
3. In the New Link Template dialog box, enter a name for the link template in New Link Template Name. Click Continue.



4 Select one or more key fields by clicking in their check boxes.

You should keep the number of key fields to a minimum. Large numbers of key fields can have a negative impact on the performance of Link Select and other connectivity operations.

5 Click OK to create the link template.

To create a link between an object and a record in an external database

1 Click Toolsmenu ➤ Palettes ➤ DbConnect.

2 In the dbConnect Manager, open a database table that has at least one link template defined.

3 In the Data View window, select the link template that you want to use from the Link Template list.

4 In the Data View window, select one or more records that you want to link.

5 Click the arrow to the right of the Link button to display the drop-down list.

6 In the drop-down list, click Create Links.

7 Select one or more objects in the drawing that you want to link the data to.

8 Press Enter.

To edit a link template

1. Click Toolsmenu ➤ Palettes ➤ DbConnect.
2. Click DbConnect menu ➤ Templates ➤ Edit Link Template.
3. In the Select a Database Object dialog box, select a link template to edit. Click Continue.
4. Select one or more key fields from the list by clicking their check boxes.
5. Click OK to save your changes to the link template.

To update a link template's properties

1. Click Toolsmenu ➤ Palettes ➤ DbConnect.
2. Click DbConnect menu ➤ Templates ➤ Link Template Properties.
3. In the Select a Database Object dialog box, select a link template. Click Continue.
4. In the Link Template Properties dialog box, update the data source information as appropriate. Click OK.

To delete all links based on a link template from the current drawing

1. Click Toolsmenu ➤ Palettes ➤ DbConnect.
2. Click DbConnect menu ➤ Links ➤ Delete Links.
3. In the Select a Database Object dialog box, select the link template whose links you want to delete. Click OK.

To edit a link's key values using the Link Manager

1. Click Toolsmenu ➤ Palettes ➤ DbConnect.
2. Click DbConnect menu ➤ Links ➤ Link Manager.
3. In the drawing, select a linked object.
4. Note that the Link Manager can work with just one graphical object at a time.
5. In the Link Manager, select a link from those available for the selected object in the Link Templates list.

6 Enter a new value in the Value field of the key field you want to edit, or click the [...] button. Repeat, if necessary, for any additional key fields that you want to edit.

The [...] button displays a list of all values for the specified field from the database table, allowing you to select the value you want. After selecting a value, click OK.

NOTE The [...] button is available only if you are connected to the data source that the selected link references.

7 Click OK to update the link and close the dialog box.

Quick Reference

Commands

DBCONNECT

Provides an interface to external database tables.

Identify and Select Existing Links

You can select linked records in the data source to automatically select the linked objects in the drawing, and you can select objects to select the linked records in the data source

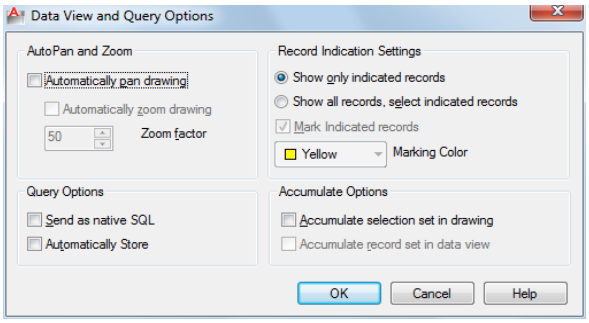
After you have created links to program graphical objects, you may want to view the data that is associated with them. For example, suppose you have linked employee database records to an office layout and you want to locate a particular employee's office in the drawing. You can select the employee's record in the Data View window to select any corresponding linked graphical objects in your drawing. Conversely, you can select objects to select any linked database records in the Data View window.

Automatic Selection of Linked Objects

You can set up the program to automatically select linked graphical objects in your drawing as additional records are selected in the Data View window. Conversely, you can set up the program to select linked database records as additional graphical objects are selected in your drawing. Only one of these automatic viewing modes can be active at a time.

Other Display Options

You can use the Data View and Query Options dialog box to set a number of viewing options that affect how linked records and linked graphical objects are displayed whenever a corresponding linked item is selected.



The Automatically Pan Drawing option automatically pans the drawing to display objects that are associated with the current selection set of records in the Data View window. If you have multiple records selected in the Data View window, AutoPan may not be able to fit all of them in the drawing window with your drawing's current zoom factor. To control for this, you can set a zoom factor for the AutoPan mode that ensures that all linked objects associated with the current record set are displayed.

The Automatically Zoom Drawing option automatically zooms the drawing so that all objects associated with the current record set are displayed. You must specify a zoom factor that sets the size of the extents of the indicated object set to a defined percentage of the drawing area. The available range is 20 percent to 90 percent, and the default value is 50 percent. A value of 50 percent means that either the height of the extents is 50 percent of the height of the drawing area, or the width of the extents is 50 percent of the drawing area, whichever is less.

You can also specify the following link viewing settings:

- **Record Indication Settings**. Specify whether all records or only the subset associated with the current selection set of graphical objects are displayed in the Data View window. If the Show All Records, Select Indicated Records option is selected, you can specify a marking color to apply to the linked records.
- **Accumulate Options**. Specify whether the program accumulates selection sets of linked graphical objects or Data View records, or creates new selection sets as additional objects or records are selected.

For a description of all the options available, see "Data View and Query Options Dialog Box" under DBCONNECT in the *Command Reference*.

To view graphical objects that are linked to a set of database records

1. In the dbConnect Manager, double-click a database table to open it.
2. In the Data View window, select a link template from the Link Template list.
3. In the Data View window, select one or more records.
4. Right-click the selected record. Click View Linked Objects

   Any objects that are linked to the selected records are selected in the drawing area.

To view database records that are linked to a selection set of graphical objects

1. In the dbConnect Manager, double-click a database table to open it.
2. In the Data View window, select a link template from the Link Template list.
3. In the drawing area, create a selection set of graphical objects.
4. In the Data View window, click the View Linked Records in Data View button.

   By default, only database table records linked to the current drawing selection set are displayed in the Data View window. You can change the settings in the Data View and Query Options dialog box to display all records, if desired.

To automatically view graphical objects that are linked to a set of database table records

1. In the dbConnect Manager, double-click a database table to open it.
2. In the Data View window, select a link template from the Link Template list.
3. In the Data View window, click the AutoView Linked Objects in Drawing button.
4. In the Data View window, select one or more records.

   Any corresponding linked graphical objects are selected in the drawing area.

5 Repeat step 4 to select additional linked graphical objects.

6 Repeat step 3 to turn off the AutoView Linked Objects mode.

To automatically view database table records that are linked to a selection set of objects

1 In the dbConnect Manager, double-click a database table to open it.

2 In the Data View window, select a link template from the Link Template list.

3 In the Data View window, click the AutoView Linked Records in Data View button.

4 In the drawing area, select one or more graphical objects.

By default, only database table records linked to the current drawing selection set are displayed in the Data View window. You can change the settings in the Data View and Query Options dialog box to display all records, if desired.

5 Repeat step 4 to select additional linked records.

6 Repeat step 3 to turn off the AutoView Linked Records mode.

To specify additional link viewing settings

1 In the Data View window, click the Data View and Query Options button.

2 In the Data View and Query Options dialog box, change the default settings as desired.

3 Click OK to save your changes and close the dialog box.

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Find and Correct Link Errors

The Synchronize option analyzes the links in your drawing and detects errors.

It is recommended that you periodically check the links in your drawings to update or delete broken links. The program provides a Synchronize option to analyze the links in a drawing that are based on a particular link template. After you run the Synchronize option, a list of detected errors is provided in the Synchronize dialog box. Certain errors (such as a resized column in the source database table) can be fixed directly from the Synchronize dialog box. Other errors (such as links that point to non-existent records) must be fixed in the source database table.

To synchronize the links in a drawing

1. In the dbConnect Manager, right-click a link template. Click Synchronize.

   If no errors are detected, a message is displayed. If there are errors, they are summarized in the Synchronize dialog box.

2. In the Synchronize dialog box, select an error and do one or more of the following:
   - Click Indicate Drawing Objects to highlight the linked graphical objects.
   - Click Fix to fix the broken link.
   - Click Delete to delete the broken link.
3. Repeat step 3 for any additional errors.
4. Click Close.

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Export Link Information

You can export information about links to a text file.

You can use the program to export links contained in a drawing. This is useful when you have multiple graphical objects linked to a single database row. Suppose, for example, that you want to prepare a bill of materials that summarizes the number of different light fixtures in a drawing. The external

database that your drawing is linked to does not contain this information. All the database provides is a list of the various light fixtures that make up its record set. The database table has no knowledge of the number of instances of individual light fixtures in your drawing. If you want to prepare a summary report containing this information, you have to export the links in your drawing.

To export links from a drawing

1. Click Toolsmenu ➤ Palettes ➤ DbConnect.
2. Click DbConnect menu ➤ Links ➤ Export Links.
3. Create a selection set of graphical objects whose links you want to export.

   If more than one link template exists for the set of graphical objects that you select, the Select Link Template dialog box is displayed and you are prompted to select a link template. If your selection set contains only one link template, the Export Links dialog box is displayed.
4. In the Export Links dialog box, select the fields to include by clicking them in the Include Fields list.

   The key fields for the selected link template are included automatically. The program also exports the entity handle of the object that each link is associated with.
5. Enter a name for the exported file in File Name.
6. Select a file format for the file from the Save As Type list.

   You may save the file in the native database format of the current table or as a comma-delimited or space-delimited text file.
7. Click Save to export the links and close the dialog box.
8. Repeat steps 1 through 6, as necessary, for any additional link templates whose links you want to export.

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Use Labels to Display Database Information in the Drawing

Labels are multiline text objects that display data in a drawing from selected fields stored in an external database table.

Create and Edit Labels and Label Templates

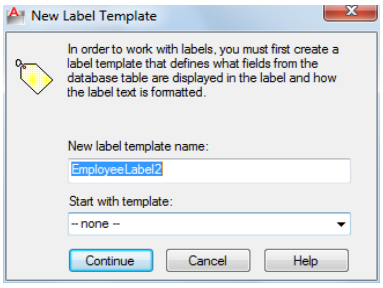
Labels may be freestanding or attached to an object. The label template defines what fields from the database table are displayed in the label.

Links provide a powerful mechanism for associating external data with program graphical objects. By selecting linked objects, you can readily access their associated records in the database table. Links do, however, have some inherent limitations. Suppose, for example, that you wanted to print a drawing and include the external data associated with individual links. Because the links are merely pointers to the external database table, the external data would not appear in your printed drawing. For those occasions when you want a visible representation of external data in a drawing, the program provides *labels.*

Labels are multiline text objects that display data in a drawing from selected fields stored in an external database table.

The labels that you create in the program can be freestanding, or they can be attached to a graphical object. Freestanding labels exist in the drawing independent of any graphical objects. Labels that are attached to a graphical object are tightly connected with it. If you move the graphical object, the label moves with it. If you copy the object to the Clipboard, the label is also copied. If you delete an object that has an attached label, the label is deleted as well. Labels associated with graphical objects are displayed with a leader.

To work with labels, you must first create a label template that defines what fields from the database table are displayed in the label and how the label text is formatted.



Edit Label Templates

After defining a label template, you might have cause to edit it. For example, you may decide that you want additional database table fields to be displayed in your drawings, or that you want to change the font or size of the label object. You can also specify a number of offset settings that affect the insertion point of labels and, in the case of attached labels, their associated leader objects.

The Label Offset tab of the Label Template dialog box specifies *X* and *Y* coordinate offset values for labels and leader objects. The following table summarizes the effect these settings have on both freestanding and attached labels.

Label Offset tab options

Option	Freestanding labels	Attached labels
Start	N/A	Specifies the start point for the leader object. The point is defined with respect to the extents of the graphical object that the label is attached to. Suppose, for example, that you specify a Start setting of Top Left for a label attached to a rectangle graphical object. The tip of the leader is inserted in the upper left corner of the rectangle.
Leader Offset	N/A	Specifies an *X* and *Y* offset for the multiline text object with respect to the associated leader object. By default, the *X* and *Y* Leader Offset options are set to 1, indicating that the label text is offset 1 unit in the *X* and *Y* planes from the tip of the leader object.
Tip Offset	Specifies an *X* and *Y* label offset from the insertion point you specify for the label. For example, *X* and	Specifies an *X* and *Y* offset for the tip of the leader object from the value specified with the Start option. Suppose, for example, that you specify a Start

Label Offset tab options

Option	Freestanding labels	Attached labels
	Y Tip Offset settings of -1 cause the label to be inserted offset 1 negative unit in the *X* and *Y* planes from the insertion point you specify in the drawing.	setting of Top Left for a label attached to a rectangle graphical object. You also specify *X* and *Y* Tip Offset values of 1. The tip of the leader is inserted offset 1 positive unit in the *X* and *Y* planes from the upper left corner of the rectangle.

Edit Labels

Once you have created labels, you can edit and manipulate them in a variety of ways. Labels and their leaders can be moved and resized using standard commands such as COPY, MOVE, and SCALE. It is important to note that changes made to individual label instances do not affect or redefine the label template.

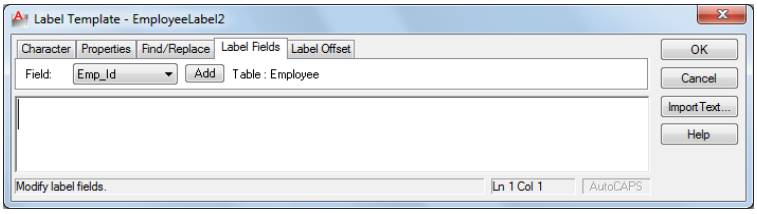
NOTE It is not recommended that you edit the text of labels using the In-place Text Editor. The new text values are not updated in the link or the database table, and original field values are restored if you click Reload. If you want to change field values for an existing label, edit the link values as described in Create and Edit Links and Link Templates on page 2015 and click Reload.

To create a label template

1. Click Toolsmenu ➤ Palettes ➤ DbConnect.
2. In the dbConnect Manager, click the New Label Template button.
3. In the New Label Template dialog box, enter a name for the label template in New Label Template Name.
4. To open the Label Template dialog box, click Continue.
5. In the Label Template dialog box, Label Fields tab, select a field to appear in the label from the Field list. Click Add to add it to the label.

   You can also enter any additional static text that you want to precede or follow the label directly in the Label Template dialog box. For example, you might create a label template that uses the field Room Number from a room database table. If you want the field name to precede the field value when the label is created, enter **Room Number:** in the Label Template dialog box before selecting the field from the Field list.
6. Repeat step 5 for any additional fields that you want displayed in your label.

7 Click OK.



To create a link with a freestanding label

1 Click Toolsmenu ➤ Palettes ➤ DbConnect.

2 In the dbConnect Manager, open a database table that has at least one link template and one label template defined.

3 In the Data View window, select a link template from the Link Template list.

4 In the Data View window, select a label template from the Label Template list.

5 In the Data View window, select the record that you want to link.

6 Right-click the record header (the empty square to the left of the row) of the selected record. Click Link And Label Settings ➤ Create Freestanding Labels.

7 Click Data View menu ➤ Link.

8 Select an area in the drawing to insert the label.

To create a link to an object with an attached label

1 Click Toolsmenu ➤ Palettes ➤ DbConnect.

2 In the dbConnect Manager, open a database table that has at least one link template and one label template defined.

3 In the Data View window, select a link template from the Link Template list.

4 In the Data View window, select a label template from the Label Template list.

5 Select the record that you want to link from the Data View window.

6 Click Data View menu ➤ Link And Label Settings ➤ Create Attached Labels.

7 Click Data View menu ➤ Link.

8 Select an object or objects in the drawing that you want the label associated with.

9 Press Enter.

To edit a label template

1 Click Toolsmenu ➤ Palettes ➤ DbConnect.

2 In the dbConnect Manager, in the tree view, right-click a label template to edit. Click Edit.

3 In the Label Template dialog box, make the changes you want as described in To create a label template.

4 Click OK.

To specify label formatting

1 Create a new label template as described in To create a label template.

2 In the Label Template dialog box, select text formatting options using the Character and Properties tabs, as described in MTEXT in the *Command Reference.*

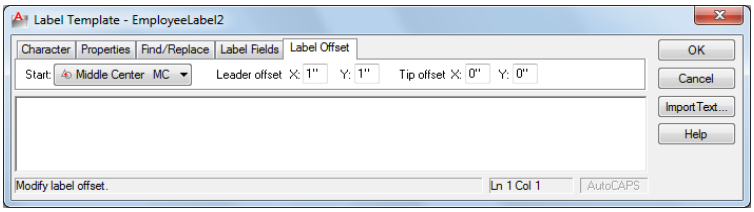
3 In the Label Offset tab, specify the insertion point for the tip of the leader object in Start.

The Start setting has no effect on freestanding labels.

4 Enter both *X* and *Y* offset values in Leader Offset to specify the offset of the label from the tip of the leader object.

By default, the *X* and *Y* offset values are set to 1, indicating that the label is offset by one drawing unit in both the *X* and the *Y* directions from the tip of the leader object.

5 Enter both *X* and *Y* offset values in Tip Offset to specify the offset for the leader object or, for freestanding labels, the offset of the label from the insertion point you specify for the label.



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Update Labels with New Values from the Database

When database records change, you update labels to reflect the new values.

After you have added labels to your drawings, you may make occasional changes to the database tables that affect the label values. If you make frequent changes to your database tables, you must regularly update the labels in your drawings to ensure data integrity.

To update all label values in the current drawing

1. Click Toolsmenu ➤ Palettes ➤ DbConnect.
2. In the dbConnect Manager, in the tree view, right-click a drawing file. Click Reload Labels.

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Use Queries to Filter Database Information

A query to the database is a search criterion that you construct to return the records you want.

Overview of Queries

Part of the fundamental power of database systems is their ability to present a subset of records based on a search criterion or *query* that you specify. For example, you might have a database table containing records of all the rooms that exist at your company's headquarters facilities. Suppose that you want to prepare a report that lists all conference rooms that can seat more than 20 people. Using the Query Editor, you can easily construct a query that returns the subset of records or linked graphical objects that you want to see.

The Query Editor consists of a series of four tabs that you can use to develop queries. The tabbed progression is designed to make working with queries simple, even if you are not familiar with Structured Query Language (SQL). If you are unfamiliar with queries, work with the Quick Query and Range Query tabs initially until you become familiar with query syntax. Once you learn the basics involved in developing meaningful queries, you can progress to the Query Builder and SQL Query tabs.

You can begin creating a query in one tab and continue to refine it or add additional parameters in subsequent tabs. You might, for example, begin creating a simple query in the Quick Query tab and subsequently decide that you would like to add an additional condition using the Query Builder tab. When you click the Query Builder tab, it displays the values that you initially selected using the Quick Query tab, and you can add additional conditions to the query. However, you cannot move backwards through the tabs once you have changed the query in one of the later tabs, because each subsequent tab provides additional functions that are not available in the previous tabs. If you attempt to move backwards through the query tabs after modifying a query, you are prompted with a warning indicating that the query will be reset with its original default values.

The tabbed progression of the Query Editor is particularly useful in becoming familiar with SQL syntax. For example, you can create a query using the Quick Query tab and then select the SQL Query tab to view how your query is formatted using SQL.

The Query Editor provides the following tabs for building queries:

- **Quick Query**. Provides an environment where you can develop simple queries based on a single database field, single operator, and single value.

For example, you can find all records from the current database table where the value of the "room type" field equals "office."

- **Range Query**. Provides an environment where you can develop a query that returns all records that fall within a given range of values. For example, you can find all records from the current database table where the value of the "room area" field is greater than or equal to 90 square feet and less than or equal to 120 square feet.
- **Query Builder**. Provides an environment where you can develop more complicated queries based on multiple search criteria. For example, you can find all records from the current database table where the "room type" equals "cubicle" and room area is greater than 80 square feet.
- **SQL Query**. Provides an environment where you can develop sophisticated queries that conform with the SQL 92 protocol. For example, you can select * from california.headquarters.room where california.headquarters.room.type = 'cubicle'

  and

  california.headquarters.room.area < 80.

See Construct Simple Queries on page 2034 for detailed information about constructing queries and working with operators. If you are already comfortable building database queries, see Use the Query Builder on page 2037.

To open the Query Editor

1. Click Toolsmenu ➤ Palettes ➤ DbConnect.
2. In the dbConnect Manager, in the tree view, select a database table.
3. Click the New Query button.
4. In the New Query dialog box, enter a name for the query in New Query Name.
5. Click Continue to open the Query Editor.

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Construct Simple Queries

The simplest query uses one operator to search one field for a certain value. The range query searches one field to return records that fall within a range you specify.

The Quick Query Tab

The Quick Query tab is designed to familiarize you with query syntax as quickly as possible. Constructing a quick query involves selecting a field from the current database table, applying a *conditional operator* to it, and specifying a value for the field.

Conditional operators are constraints such as equal, greater than, or less than. The following table summarizes the Quick Query tab's operators.

Query Operators

NOTE All operators can be applied to both numeric and text-based fields. For example, you can search for all records greater than "c", which would return all records ranging from "ca..." through "z...". It is important to realize that query searches are case sensitive: "ROOM" is not equivalent to "room".

The Range Query Tab

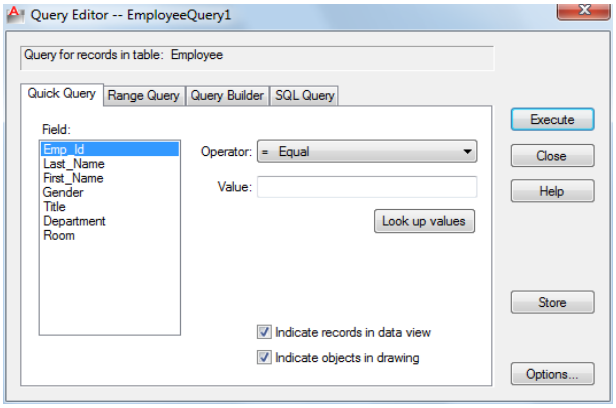
Like the Quick Query tab, the Range Query tab is designed to assist users who are new to working with database queries. Using the Range Query tab, you can specify a range of values for the query to return: for example, all rooms that are greater than or equal to 80 square feet but are less than or equal to 100 square feet. The interface is similar to that of the Quick Query tab, except the Operator field has been removed and the Value field has been replaced with two fields: a From field and a Through field. All other interface features are available and work as they do on the Quick Query tab.

To open the Query Editor

1. Click Toolsmenu ➤ Palettes ➤ DbConnect.
2. In the dbConnect Manager, in the tree view, select a database table.
3. Click the New Query button.
4. In the New Query dialog box, enter a name for the query in New Query Name.
5. Click Continue to open the Query Editor.

To construct a quick query

1 Open the Query Editor.



2 On the Quick Query tab, select a field from the Field list.

3 Select an operator from the Operator list.

4 Enter a value in the Value field, or click Look Up Values.

Look Up Values returns a list of all values for the specified field from the database table, so you can select the value that you want. After selecting a value, click OK.

5 Select one or both of the following:

- **Indicate Records in Data View**. Highlights records that match your query in the Data View window.
- **Indicate Objects in Drawing**. Highlights objects that match your query in the current drawing.

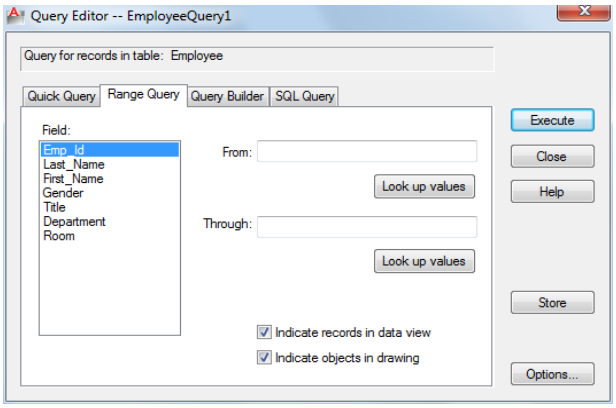
6 If you want to save the query in the current drawing for future use, click Store.

7 Click Execute to execute the query and close the dialog box.

A subset of records that match your query criterion are displayed in the Data View window. To return to the Query Editor to refine your query, click the Return to Query button.

To construct a range query

1 Open the Query Editor.



2 On the Range Query tab, select a field from the Field list.

3 Enter a value in From, or click Look Up Values (From).

Look Up Values displays a list of all values for the specified field from the database table, so you can select the value that you want. After selecting a value, click OK.

The query searches for all values greater than or equal to the value that you specify.

4 Enter a value in Through, or click Look Up Values (Through).

The query searches for all values less than or equal to the value that you specify.

5 If you want to save the query in the current drawing for future use, click Store.

6 Click Execute to execute the query and close the dialog box.

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Use the Query Builder

Boolean operators and parenthetical grouping are used to create queries with two or more search criteria.

The Query Builder tab is the primary query environment. Unlike the Quick Query and Range Query tabs, the Query Builder can be used to create queries based on multiple criteria. You can also parenthetically group criteria and specify what fields are displayed and how they are sorted when the query is returned in the Data View window.

The Query Builder tab introduces *Boolean operators* and parenthetical grouping, which are used to create compound queries based on two or more search criteria. For example, using Boolean operators, you can generate a query that returns a list of all light fixtures of a given type that were installed after a specific date.

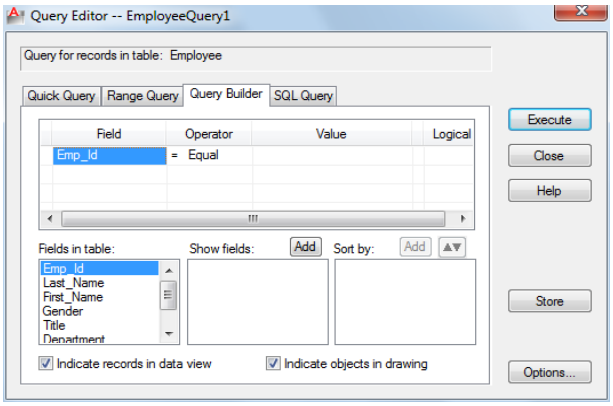
- **And Operator**. Constructs a query based on multiple criteria, returning a set of records that meet all specified criteria. For example, search for all rooms that are offices and are currently unoccupied. The query returns records for all rooms that are *both* offices and unoccupied.
- **Or Operator**. Constructs a query based on multiple criteria, returning a set of records that meet any of the specified criteria. For example, search for all rooms that are offices or are currently unoccupied. The query returns records for all rooms that are *either* offices or unoccupied.
- **Parenthetical Grouping**. Groups a series of search criteria by bracketing them within parentheses. For example, search for
  (All rooms that are offices and are currently unoccupied)

  or

  (All rooms that are cubicles and are currently unoccupied)

  This query returns a list of all unoccupied offices and cubicles.

  You can nest up to four sets of parentheses within a statement.



To open the Query Editor

1. Click Tools menu ➤ Palettes ➤ dbConnect.
2. In the dbConnect Manager, in the tree view, select a database table.
3. Click the New Query button.
4. In the New Query dialog box, enter a name for the query in New Query Name.
5. Click Continue to open the Query Editor.

To construct a query with a single parameter on the Query Builder tab

1. Open the Query Editor.
2. On the Query Builder tab, select a field from the drop-down list in the Field cell.
3. Select an operator from the Operator cell drop-down list.
4. Enter a value in the Value cell, or click the [...] button.

   The [...] button returns a list of all values for the specified field from the database table, allowing you to select the value you want. After selecting a value, click OK.
5. If you want to save the query in the current drawing for future use, click Store.
6. Click Execute to execute the query and close the dialog box.

To construct a query with multiple parameters on the Query Builder tab

1. Enter the first parameter of your query as described above.
2. Select the appropriate And or Or operator from the Logical field.

   Clicking in the Logical field inserts an And operator. Clicking the cell again changes the value to Or.
3. Enter additional query parameters on subsequent rows of the Query grid.
4. Repeat step 2 for each additional parameter except the final query parameter.
5. Apply any parenthetical grouping, if appropriate. Click the cell to the left of the Field cell that begins the phrase that you want to group. A single left parenthesis is inserted. Click the cell to the right of the Value cell that ends the phrase that you want to group. A single right parenthesis is inserted.
6. If you want to save the query in the current drawing for future use, click Store.
7. Click Execute to execute the query and close the dialog box.

To specify what fields appear in your returned query

1. Construct your query as described above.
2. Select the first field that you want to appear in the Fields in Table list.
3. Above the Show Fields list, click Add.
4. Repeat steps 2 and 3 for any additional fields you want to display.

To specify a sort order for your returned query

1. Construct your query as described above.
2. Select the first field that you want to sort on from the Show Fields list.
3. Above the Sort By list, click Add.

   By default, the selected field is sorted in ascending order. Click the Sort button to sort in descending order.
4. Repeat steps 2 and 3 for any additional fields that you want to sort on.

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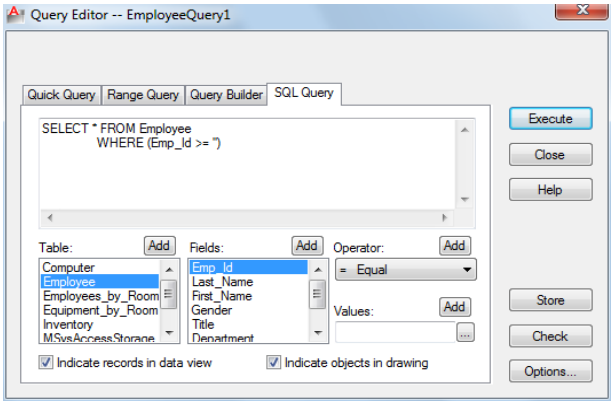
Provides an interface to external database tables.

Use SQL Queries

If your database is fully compliant with the SQL 92 standard, you can use SQL statements to query more than one database table at a time.

The SQL Query tab provides you with both a query editor box where you can enter a free-form SQL query and a set of tools to assist you in constructing a query. You can issue any valid SQL statement that conforms with Microsoft's implementation of the SQL 92 protocol from this tab. Because not all database management systems are fully compliant with the SQL 92 standard, you should review your system's documentation to see what SQL commands are valid for your particular database.

Unlike the other tabs in the Query Editor, the SQL Query tab can be used to construct a query that performs relational operations on multiple database tables using the SQL join operator. Because joins are a relatively advanced SQL operation, refer to your database system documentation or to the SQL 92 protocol for information about working with joins.



The following additional interface elements are introduced on the SQL Query tab:

- **Table List**. Lists all database tables available in the current data source. You can add tables to the SQL query editor by double-clicking them, by selecting them and choosing Add, or by typing their names directly in the SQL query editor.
- **Check**. Checks your SQL query for proper syntax without actually executing it. This function helps you isolate syntax errors before you issue your query.

To construct an SQL query

1. Open the Query Editor as described in To open the Query Editor.
2. Click the SQL Query tab.
3. Enter your query in the Query Editor, or select query components from the interface as described in Use the Query Builder on page 2037.
4. If you want to save the query in the current drawing for future use, click Store.
5. Click Execute to execute the query and close the dialog box.

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Combine Queries

An advanced type of query uses operators on successive selection sets to return a new set of objects or records, which then can be further refined by the same process.

Link Select is an advanced implementation of the Query Editor that constructs iterative selection sets of graphical objects and database records.

Link Select creates an ongoing selection set that you refine with additional information through a running iterative process. You begin either by constructing a query or by selecting graphical objects. This initial selection set is referred to as set A. You can now select additional graphical objects or

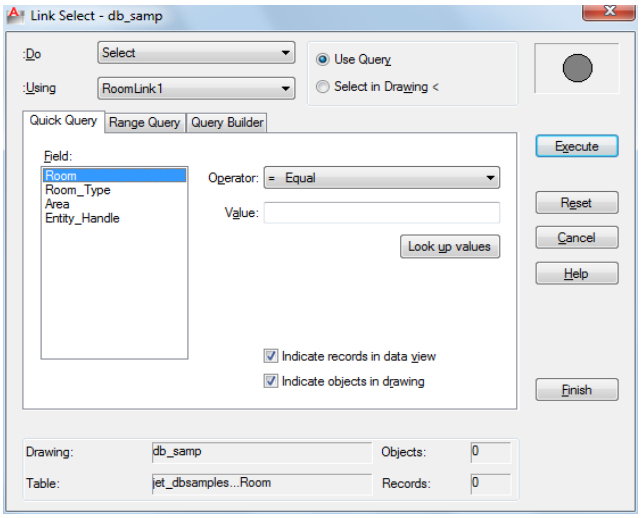
issue a query to further refine your selection set. This second selection set is referred to as set B. You must establish a relationship between set A and set B to further refine the selection set. The following relationships or set operations are available:

- **Select.** Creates an initial query or selection set. This selection set can be refined through subsequent Link Select operations.
- **Union.** Adds the results of the new query or selection set to the running selection set. This operation returns all records or objects that are members of set A *or* set B.
- **Intersect.** Returns the intersection of the existing running selection and the results of the new query or drawing selection. This operation returns all records or objects that are members of set A *and* set B.
- **Subtract A - B.** Subtracts the results of the new query or drawing selection from the existing running selection.
- **Subtract B - A.** Subtracts the existing running selection from the results of the new query or drawing selection.

After you apply one of the set operations, the returned results become the new running selection and are assigned to set A. You can continue the iterative process by creating additional B sets to further refine your Link Select set.

To use Link Select

1. Click Toolsmenu ➤ Palettes ➤ DbConnect.
2. In the dbConnect Manager, in the tree view, right-click a link template. Click Link Select.



3 Select the Select option from the Do list.

4 Select a link template from Using.

5 Select one of the following options:

- **Use Query.** Constructs a query using one of the tabs in the Query Editor.
- **Select in Drawing.** Closes the dialog box temporarily so that you can select graphical objects in the drawing. When you select the Select in Drawing option, the Execute button changes to Select.

6 Click either Execute or Select to add your query or graphical object selection set to the Link Select operation.

7 Select a Link Select operation from the Do list: Union, Intersect, Subtract A-B, or Subtract B-A.

8 Repeat steps 3 through 5 to add set B to the Link Select operation.

9 Select one or both of the following options and click Finish to complete the Link Select operation:

- **Indicate Records in Data View.** Returns the results of the current Link Select operation in the Data View window when you click Finish. You can perform further Link Select iterations by clicking the Return to Link Select button in the Data View window.

- **Indicate Objects in Drawing.** Creates a selection set of linked graphical objects in the drawing area when you click Finish. You can perform further Link Select iterations by pressing Enter at the command prompt.

NOTE It is recommended that you turn off the Indicate Records in Data View option if you are performing a Link Select operation on links from more than one database table. Link Select only displays records from the table that the current link template references. This limitation can cause misleading results to be returned to the Data View window.

Quick Reference

Commands

DBCONNECT

Provides an interface to external database tables.

Save and Reuse Queries

When you save a query, you can later edit it, rename it, and copy it into other drawings.

You can save queries that you plan on using more than once with a drawing. Queries that you save are listed in the dbConnect Manager under the drawing node of the drawing that they were created in. You can edit and rename stored queries or copy them to other drawings.

To execute a stored query

1. Click Toolsmenu ➤ Palettes ➤ DbConnect.
2. In the dbConnect Manager, in the tree view, right-click a stored query. Click Execute.

To edit a stored query

1. Click Toolsmenu ➤ Palettes ➤ DbConnect.
2. In the dbConnect Manager, in the tree view, right-click a stored query. Click Edit.

3 Use the Query Editor dialog box to make the necessary changes. Click Store to save the edited query with the current drawing.

To rename a stored query

1 Click Toolsmenu ➤ Palettes ➤ DbConnect.

2 In the dbConnect Manager, right-click the query you want to rename. Click Rename.

3 Enter a new name for the query. Press Enter.

To delete a stored query

1 Click Toolsmenu ➤ Palettes ➤ DbConnect.

2 In the dbConnect Manager, in the tree view, right-click a stored query. Click Delete.

To copy a stored query to another drawing

1 Click Toolsmenu ➤ Palettes ➤ DbConnect.

2 In the dbConnect Manager, select the query that you want to copy.

3 Drag the query to the node of the drawing that you want to copy it to.

To automatically store new queries with the current drawing

1 Click Toolsmenu ➤ Palettes ➤ DbConnect.

2 In the dbConnect Manager, in the tree view, open a database table by double-clicking it.

3 In the Data View window, click the Query button.

4 In the New Query dialog box, click Continue.

5 In the Query Editor dialog box, click Options.

6 In the Data View and Query Options dialog box, Query options area, select Automatically Store. Click OK.

Quick Reference

Commands

DBCONNECT

Provides an interface to external database tables.

Share Link and Label Templates and Queries with Other Users

You can export and import links and link templates, labels and label templates, and queries.

Occasionally you may want to make templates or queries that you have developed available to other users, or use templates or queries that others have developed. This is particularly useful when developing a set of common tools that can be shared by all of the team members on a project, and it eliminates the need to replicate the work of template and query development on each team member's workstation. The program can import and export all the link and label templates and queries that are associated with a drawing.

To import a set of templates into the current drawing

1. Click Toolsmenu ➤ Palettes ➤ DbConnect.
2. In the dbConnect Manager, in the tree view, right-click a drawing file. Click Import Template Set.
3. In the Import Template Set dialog box, select a template set.
4. Click Open to import the template set into the current drawing.

   If there is a link or label template with the same name already associated with the current drawing, an alert box is displayed that you can use to provide a unique name for the template.

To export a set of templates from the current drawing

1. Click Toolsmenu ➤ Palettes ➤ DbConnect.
2. In the dbConnect Manager, in the tree view, right-click a drawing file. Click Export Template Set.
3. In the Export Template Set dialog box, in the Save In list, select the directory you want to save the template set to.

4 Under File Name, specify a name for the template set. Click Save.

To import a set of queries into the current drawing

1 Click Toolsmenu ➤ Palettes ➤ DbConnect.

2 In the dbConnect Manager, in the tree view, right-click a drawing file. Click Import Query Set.

3 In the Import Query Set dialog box, select the query set.

4 Click Open to import the query set into the current drawing.

If there is a query with the same name already associated with the current drawing, an alert box is displayed that you can use to provide a unique name for the query.

To export a set of queries from the current drawing

1 Click Toolsmenu ➤ Palettes ➤ DbConnect.

2 In the dbConnect Manager, in the tree view, right-click a drawing file. Click Export Query Set.

3 In the Export Query Set dialog box, in the Save In list, select the directory where you want to save the query set.

4 Under File Name, specify a name for the query set. Click Save.

Quick Reference

Commands

DBCONNECT

Provides an interface to external database tables.

Work with Links in Files from Earlier Releases

You can convert ASE links in drawings from Releases 12, 13, and 14 to AutoCAD 2000 or later format. You can save AutoCAD 2000 links in Release 13 and Release 14 format.

Because AutoCAD 2000 and later releases store links in a different format than previous releases, you need to convert your links if you plan on using them with AutoCAD 2000 and later. Note the changes in terminology: *key value*

replaces *key column* in Release 14; *link template* replaces *link path name;* and *label* replaces *displayable attribute*. You also need to create a configuration file that points to the data source referenced by the legacy links.

When you open a drawing that contains legacy links, the program attempts to perform an automatic conversion of the legacy information. If, for example, you open a Release 14 drawing that contains links that point to a data source with the structure *db3.ase.db3sample.computer(com)*, the program attempts to establish an exact, one-to-one mapping of the data source components.

In some cases, the legacy data source might not match exactly how the data source is configured. For example, the legacy data source might specify a catalog that isn't specified in the data source. If the automatic link conversion is unsuccessful, you can use the Link Conversion dialog box to establish a mapping between the legacy and new data sources. You can also use the Link Template Properties dialog box to specify new data source parameters for a selected link template.

During the conversion process, the data source mapping information is written to a file called *asi.ini*. You can manually edit the conversion mapping information specified in this file, if necessary.

NOTE The program attempts to automatically convert legacy links only for data sources that don't have mapping values specified in the *asi.ini* file. Once you have specified conversion mappings for a particular data source, the program uses the settings specified in the *asi.ini* file for any additional legacy drawings you open that contain links that reference this data source.

Save Links to Earlier Formats

Links that you have created in AutoCAD 2000 and later can be converted to Release 13 or Release 14 links. AutoCAD 2000 and later links cannot be converted to Release 12 format. When you use SAVEAS to save a drawing in the current format that contains links to Release 13 or Release 14 format, the links are automatically converted to the appropriate drawing format. You must specify a conversion mapping as described above.

To convert Release 13 and Release 14 links

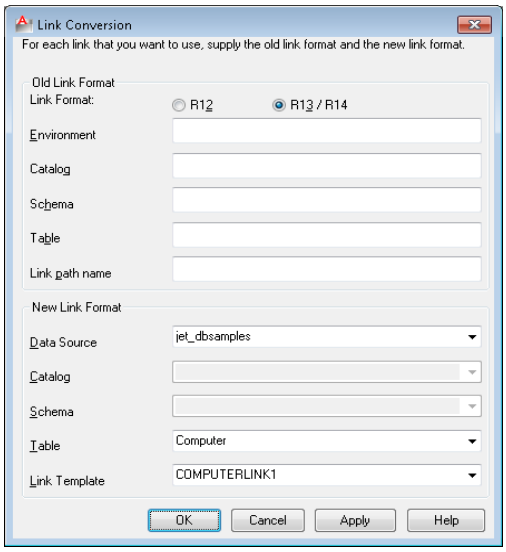
1. Click Toolsmenu ➤ Palettes ➤ DbConnect.
2. Click DbConnect menu ➤ Link Conversion.
3. In Old Link Format, do *all* of the following:
   - Select R13/R14 Link Format.

- In Environment, enter the name of the Release 13 or Release 14 environment for the link that you want to convert.
- In Catalog, enter the name of the Release 13 or Release 14 catalog for the link that you want to convert.
- In Schema, enter the name of the Release 13 or Release 14 schema for the link that you want to convert.
- In Table, enter the name of the Release 13 or Release 14 database table for the link that you want to convert .
- In Link Path Name, enter the link path name for the link that you want to convert.

4 In New Link Format, do *all* of the following:
- Select the appropriate data source from the Data Source list.
- Select the appropriate catalog from the Catalog list.
- Select the appropriate schema from the Schema list.
- Select the appropriate table from the Table list.
- Select the appropriate link template from the Link Template list.

5 Click OK.

6 Open the drawing whose links you want to convert, and save it.



NOTE You must specify an environment, database table, and link path name for all links that you convert. Catalog and schema are optional parameters that are not required by all database management systems.

To convert Release 12 links

1. Click Toolsmenu ➤ Palettes ➤ DbConnect.
2. Click DbConnect menu ➤ Link Conversion.
3. In Old Link Format, do *all* of the following:
   - Select R12 Link Format.
   - In DBMS, enter the name of the Release 12 database management system for the link that you want to convert.
   - In Database, enter the name of the Release 12 database for the link that you want to convert.
   - In Table, enter the name of the Release 12 database table for the link that you want to convert.
4. In New Link Format, do *all* of the following:
   - Select the appropriate data source name from the Data Source list.

- Select the appropriate catalog from the Catalog list.
- Select the appropriate schema from the Schema list.
- Select the appropriate table from the Table list.
- Select the appropriate link template from the Link Template list.

5 Click OK.

6 Open the drawing whose links you want to convert, and then save it.

Once you have specified a link conversion mapping, when you subsequently open and save in AutoCAD 2000 or later format a drawing that contains links that are specified in the mapping, the links are converted to the new format.

Quick Reference

Commands

DBCONNECT

Provides an interface to external database tables.

Manage Files with Autodesk Vault

Autodesk Vault gives you more power to manage files and track changes. Versioned copies of master files are maintained, allowing you to easily revert to earlier versions of files.

For information about using the Vault, refer to the Vault Help system.

NOTE The main components for the Autodesk Vault can be downloaded from the Autodesk Subscription site.

See also:

- Access Subscription Center

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Part 12: Collaborate with Others

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Protect and Sign Drawings 40

You can provide a safe environment for sending and receiving data, and maintain the authenticity of drawings. A password protects a drawing with encryption, and a digital signature identifies an individual or an organization through a digital ID (certificate).

Drawing Encryption

When a password is attached to a drawing, the password encrypts the drawing and prevents unauthorized people from viewing it.

Overview of Encryption

Password-based encryption helps secure drawing data when you collaborate on projects. Specifically, when you maintain a password for a drawing, you can help prevent unauthorized viewing of the drawing when it is sent to others.

When a password is attached to a drawing, the password encrypts the drawing and prevents unauthorized people from viewing it. You can choose the level of encryption used. Passwords help you collaborate on projects by helping to keep your data safe when the data is sent to others.

The drawing encryption password is different from other types of drawing passwords, such as the password you use with eTransmit, because you can specify the encryption level.

The primary use of encryption is to help prevent drawing data from being stolen. Encryption can also be used to help keep data confidential.

For example, if a company creates a valuable, confidential design for an engine factory, the company can help protect the drawings with a password (or phrase) before sending the drawings out for review by the builders. Password protection could be one of the many measures the company could take, in addition to

measures such as a secure intranet and rules about information disclosure, that would help keep the factory design confidential.

Passwords work only on AutoCAD 2004® and later drawing files (DWG, DWS, and DWT files).

NOTE If you want to attach a password and a digital signature to a drawing file, attach the password first. Modifications to files, including the adding of passwords, invalidate their digital signatures.

Quick Reference

Commands

SECURITYOPTIONS

Specifies password or digital signature options for a drawing file.

Protect Drawings with Encryption

If you add a password to a drawing and save the drawing, the drawing is encrypted and cannot be reopened until the password is entered.

Add a Password to a Drawing

A password can be a single word, or a phrase. It is recommended that you use common characters. The more characters you use, the harder it is for an unauthorized program, or "password cracker," to determine the password.

When you add a password to a drawing, you can choose whether to encrypt drawing properties, and you can specify an encryption type and key length.

After you add a password to a drawing and save the drawing, the password is required before any program or part of a program can read the data in the drawing, even if the drawing is an external reference. A specific password generally is required to be entered once per session.

You can attach a password to a file while modifying the file or when you save the file. You must individually add a password to each drawing you want to encrypt.

WARNING If you lose a password, your drawing file is lost. Before you add a password to a drawing, you should create a backup that is not protected with a password.

If you encrypt drawing properties, such as the title, author, subject, and keywords, then a password is required to view the properties and thumbnail preview of the drawing. If you decide to specify an encryption type and key length, you can select them from the ones available on your computer.

NOTE Settings for a password apply only to the current drawing.

To add a password while modifying a drawing

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Open and Save tab, click Security Options.
3. In the Security Options dialog box, Password tab, enter a password.
4. To encrypt drawing properties, such as the title, author, subject, and keywords, click Encrypt Drawing Properties.
5. Click OK.
6. In the Confirm Password dialog box, enter the password you used. Click OK.

WARNING If you lose a password, it is not recoverable. Before you add a password to a drawing, you should create a backup that is not protected with a password.

To add a password before saving a drawing

1. Click File menu ➤ Save As.
2. In the Save Drawing As dialog box, before saving the file, click Tools menu ➤ Security Options.
3. In the Security Options dialog box, Password tab, enter a password.
4. To encrypt drawing properties, such as the title, author, subject, and keywords, click Encrypt Drawing Properties.
5. Click OK.
6. In the Confirm Password dialog box, enter the password you used. Click OK.

WARNING If you lose a password, it is not recoverable. Before you add a password to a drawing, you should create a backup that is not protected with a password.

To remove a password from a drawing

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Open and Save tab, click Security Options.
3. In the Security Options dialog box, Password tab, clear the Password or Phrase to Open This Drawing option.
4. Click OK.

Quick Reference

Commands

SECURITYOPTIONS

Specifies password or digital signature options for a drawing file.

Choose the Type of Encryption

You can choose an advanced encryption level to protect your drawing.

A default encryption level, supplied by your operating system, is used unless you choose an advanced level. If you decide to choose an advanced level, you can choose from RC4 encryption providers supplied by your operating system. You also can choose a key length (the higher the length, the higher the protection). An RC4 encryption provider is a standard form of encryption, and the RC4 encryption providers vary in the type of protection they provide.

Before using an advanced encryption level (provider, type, and key length), confirm that any computer that needs to decrypt the drawing has the cryptography service provider you plan to use. For more information on cryptography providers, encryption types, and encryption key lengths, search the Internet for these terms.

To choose the encryption provider and key length for a drawing

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Open and Save tab, click Security Options.
3. In the Security Options dialog box, Password tab, enter a password or phrase to open the drawing. Click Advanced Options.
4. In the Advanced Options dialog box, choose an encryption provider, key length, or both. Click OK.

5 In the Security Options dialog box, click OK.

Quick Reference

Commands

SECURITYOPTIONS

Specifies password or digital signature options for a drawing file.

View Password-Protected Drawings

To view the data in a password-protected drawing, you must first obtain and enter the password. After you enter the password, it stays with the drawing, even if you modify and save the drawing, unless you remove the password.

Enter a Password to View a Drawing

To view data in a password-protected drawing, you open the drawing in a standard way and enter the password. Passwords are not case-sensitive.

Unless a drawing's properties, such as the title, author, subject, and keywords, were encrypted when the password was attached, you can view the properties in Properties dialog box in Microsoft® Windows Explorer.

A password-protected drawing may contain external references (xrefs) to password-protected drawings. If you open a password-protected drawing that contains xrefs, you are prompted to enter the password for the current drawing. Then you are prompted to enter the passwords of any xrefs whose passwords are different from a password you have entered in the current session. A specific password that applies to a drawing or xref generally is required to be entered once per session.

A drawing and its external references remain password-protected until the passwords are removed. You generally should remove passwords before performing batch operations on files, unless you are signing files.

If data in a password-protected drawing is read for any reason by the program, the password must be entered at least once per session of program use.

To view a password-protected drawing

1 Click File menu ➤ Open.

2 In the Select File dialog box, select a file. Click Open.

3 In the Password dialog box, enter a password.

4 Click OK.

Quick Reference

Commands

SECURITYOPTIONS

Specifies password or digital signature options for a drawing file.

View Password-Protected Xrefs

You can view drawings that contain xrefs to password-protected drawings.

If you open a password-protected drawing that contains xrefs, you must enter a password for any xref whose password is different from the current drawing's password.

You are only prompted to enter passwords that you have not yet entered. For example, if an xref shares a password with a drawing you have accessed in the current program session, such as an open drawing or an xref, you do not need to continue entering the same password.

To view a drawing with password-protected xrefs

1 Click File menu ➤ Open.

2 In the Select File dialog box, select a file. Click Open.

3 If prompted to enter passwords in the Password dialog box, enter any necessary passwords. Click OK.

Quick Reference

Commands

SECURITYOPTIONS

Specifies password or digital signature options for a drawing file.

View Properties of an Encrypted Drawing

You can view drawing properties, such as the title, author, subject, and keywords, unless they were encrypted when a password was attached to the drawing.

If drawing properties were not encrypted when a password was attached, you can view the properties in your operating system's Properties dialog box.

To view properties of an encrypted drawing

1. In Windows Explorer, locate the file whose properties you want to view.
2. Right-click the file name. Click Properties.

Quick Reference

Commands

SECURITYOPTIONS

Specifies password or digital signature options for a drawing file.

Sign Drawings

Digital signatures maintain authenticity and provide a safe environment for sending and receiving data. You can work on collaborative projects, transmit files over the Internet, and help guarantee that the files have not been altered.

Overview of Digital Signatures

With a digital signature, you can collaborate more easily with others on projects. Recipients of drawings are provided with reliable information about who created a set of drawings and whether they were modified since they were digitally signed.

Specifically, digital signatures provide the following benefits:

- Recipients of digitally signed files can be sure that the organizations or individuals who sent the files are who they claim to be.
- A digital signature guarantees that a file has not changed since the file was signed.

- A signed file cannot be rejected as invalid. The signer of a file cannot disown the file later by claiming the signature was forged.

A digital signature is not the same as a digitized signature. While a digital signature helps prove your identity and a drawing's authenticity, a digitized signature is nothing more than an electronic version of your own signature. It can be forged and copied, and has no real security value.

Digital IDs use two keys—a *public key*, which anyone can use to validate a digital signature, and a *private key*, which is known only by the originator of the digital ID. The private key is used to create the digital signature. To learn more about digital IDs, digital signatures, and public and private keys, search the Internet for these terms.

You can attach signatures to files that are compatible with AutoCAD 2000 and later file formats, including files generated by the eTransmit feature. To attach a signature to multiple files at once, or to attach a signature to a file compatible with the AutoCAD 2000 or later drawing-file format, see "Attach Digital Signatures Dialog Box" in the *Command Reference*.

NOTE Once you attach a digital signature to a drawing file, it will either be valid or invalid. You cannot remove the digital signature from the drawing file. If a digital signature was attached to a drawing in error, save the drawing in DXF format, open the DXF file, and then save it in DWG format.

Quick Reference

Commands

SECURITYOPTIONS

Specifies password or digital signature options for a drawing file.

SIGVALIDATE

Displays information about the digital signature attached to a drawing file.

System Variables

SIGWARN

Controls whether a warning is presented when a file with an attached digital signature is opened.

Utilities

Attach Digital Signatures

Attaches a digital signature to files.

Personally Sign Drawings

When you attach a digital signature to a file, anyone who views the file is notified if modifications were made after you signed it. Modifications invalidate a digital signature.

Obtain a Digital ID

To attach a digital signature to a file, you must have a digital ID (certificate), which is issued by a certificate authority. A digital ID identifies either an individual or an organization.

A digital ID contains a name, serial number, expiration date, and other information that certifies the digital signature. From a certificate authority, you can obtain digital IDs with a security level of Low, Medium, or High.

- **Low.** Use a Low security level if you want to sign multiple files quickly. A digital signature is automatically attached to all file types that are valid for a digital signature.
- **Medium.** Use a Medium security level if you want to be informed when an application is trying to create a signature with your digital ID. You are notified each time a digital signature containing your digital ID is attached to a file.
- **High.** Use a High security level if the signature is very sensitive and you do not want your computer to be misused in the signing of a file. You are prompted for a password each time you sign a file.

You can set an option to automatically attach your signature to a drawing whenever you save it. You also can sign many drawings at once, in a batch, and you can sign transmittal packages of drawings.

NOTE To prevent changes to the drawing, applying a digital signature to a drawing automatically sets the FIELDEVAL system variable to 0 (manual update).

To obtain a digital ID on the Internet

1. Open a search engine in your Internet browser and search for the term "digital certificate."
2. Click a search result to find out more about a digital ID vendor and how to obtain a digital ID.

Quick Reference

Commands

SECURITYOPTIONS

Specifies password or digital signature options for a drawing file.

SIGVALIDATE

Displays information about the digital signature attached to a drawing file.

System Variables

SIGWARN

Controls whether a warning is presented when a file with an attached digital signature is opened.

Utilities

Attach Digital Signatures

Attaches a digital signature to files.

Attach a Signature to a Single File

When you attach a digital signature to a file, you are helping to ensure that recipients of the file are notified of modifications.

You can set an option for a signature to be attached after you save a file. A single digital signature can be attached per file.

If you plan to protect a set of drawings with a password and a signature, first add the passwords, and then sign the files in a batch. Adding a password to a signed drawing, or otherwise changing a signed drawing, invalidates the signature.

NOTE Unless you use the Attach Digital Signatures utility, you must individually sign each drawing.

To set an option for a signature to be attached after you save a file

1. Do one of the following:
   - Click Tools menu ➤ Options. In the Options dialog box, Open and Save tab, click Security Options.
   - Click File menu ➤ Save As. In the Save Drawing As dialog box, click Tools ➤ Security Options.
2. In the Security Options dialog box, Digital Signature tab, select Attach Digital Signature After Saving Drawing.
3. Click OK.

   NOTE In the Security Options dialog box, you can attach digital signatures only to files that are compatible with the AutoCAD 2000 and later drawing-file formats.

Quick Reference

Commands

SECURITYOPTIONS

Specifies password or digital signature options for a drawing file.

SIGVALIDATE

Displays information about the digital signature attached to a drawing file.

System Variables

SIGWARN

Controls whether a warning is presented when a file with an attached digital signature is opened.

Utilities

Attach Digital Signatures

Attaches a digital signature to files.

Attach Digital Signatures to Multiple Files

When you attach a digital signature to a set of files, you are helping to ensure that anyone viewing the files knows about any changes that are made to the files after you signed them.

You can attach a digital signature to the following file types:

- EXE and ZIP files generated by eTransmit
- DWS files
- DWG and DWT files that are compatible with AutoCAD 2000 and later drawing-file formats

When you attach a digital signature to a set of files, you can select the files individually from within the signing utility or drag them into the utility from Windows Explorer.

To attach a digital signature to multiple files

1. Click Start menu (Windows), ➤ All Programs ➤ Autodesk ➤ AutoCAD 2012 ➤ Attach Digital Signatures.
2. In the Attach Digital Signatures dialog box, click Add Files.
3. In the Select File dialog box, select the files you want to attach a digital signature to. Click Open.

   NOTE In the Attach Digital Signatures dialog box, you can attach digital signatures to files that are compatible with AutoCAD 2000 and later drawing-file formats.

4. In the Attach Digital Signatures dialog box, in the Select a Digital ID (Certificate) box, select a digital ID to attach to the files you selected in the Files to Be Signed list.
5. In the Get Time Stamp From box, select a time service if you want to add a time stamp to the signed files. If you do not want to add a time stamp, use the default option No Time Stamp.
6. In the Comment box, add any information that is relevant to the files you are signing.
7. Click Sign Files.

8 If the file you want to sign is read-only, the Read-only File dialog box is displayed. Do one of the following:

- To sign a read-only file, click Yes.
- To sign all read-only files, select Use the Same Answer for All Read-only Files in This Batch. Click Yes.
- To cancel the application of the digital signature to a single read-only file, click No.
- To cancel the application of the digital signature to all of the read-only files, click Cancel.

NOTE If you chose either a Medium or High security level when you obtained your digital ID, a message is displayed each time you try to attach a digital signature to a file.

9 When all files have been signed, the Signing Complete message displays the number of files that were signed. Click OK.

Files that were successfully signed display the text "Signed" in the Status column of the Files to Be Signed list.

10 Click Close.

NOTE When you sign multiple files at once, the files become read-only files to help prevent someone from accidentally invalidating a digital signature.

Quick Reference

Commands

SECURITYOPTIONS

Specifies password or digital signature options for a drawing file.

SIGVALIDATE

Displays information about the digital signature attached to a drawing file.

System Variables

SIGWARN

Controls whether a warning is presented when a file with an attached digital signature is opened.

Utilities

Attach Digital Signatures

Attaches a digital signature to files.

Add a Comment and Time Stamp

You can add a comment and time stamp to a digital signature.

Servers are used to obtain the current time when adding a time stamp to a digital signature. When you send drawing files over the Internet or work on collaborative projects, you can use a specific server to create an accurate, consistent time stamp.

Several government operations (such as the National Institute of Standards and Technology, or the US Naval Observatory) and public systems (such as California Institute of Technology) offer accurate time values. You can also use your own server to create a time stamp by modifying the *timesrvr.txt* file or creating a custom text file that contains the servers you want to use when adding a time stamp to a digital signature.

The *timesrvr.txt* file is located in the program's install directory. By default the install directory for the program is located at \Program Files\AutoCAD 2012. To add a new server, open the *timesrvr.txt* file with a text editor like Notepad and add the required information for the server. The information that you need to add to the *timesrvr.txt* file when you want to added your own server is a description and the name of the server. The description that is provided is the value that appears in the drop-down and the name of the server is the URL (Uniform Resource Locator) to the server. The information for each server listed in the *timesrvr.txt* file must be in the following format:

Description (server_name)

For example:

National Institute of Standards and Technology (time.nist.gov)

NOTE Each server that you add needs to be on a separate line in the text file used.
.

To use a file different from the *timesrvr.txt* file that contains the available time servers to use when creating a time stamp for a digital signature, change the setting of the ADSKSIGTIMESRVR environment variable. For example, change the variable to *c:\temp\mytimelist.txt*

To choose a time server and add a comment

1. Do one of the following:
   - Click Tools menu ➤ Options. In the Options dialog box, Open and Save tab, click Security Options.
   - Click File menu ➤ Save As. In the Save Drawing As dialog box, click Tools ➤ Security Options.
2. In the Security Options dialog box, Digital Signature tab, select Attach Digital Signature After Saving Drawing.
3. In the Get Time Stamp From box, click a time server.
4. In the Comment box, add a comment.
5. Click OK.

To add a time server to the Time Server list

1. Locate and open the *timesrvr.txt* file.

   The *timesrvr.txt* file contains time service information.
2. In the *timesrvr.txt* file, add the time service you want to use. Use the following format:

   Description (server_name)

   For example:

   National Institute of Standards and Technology (time.nist.gov)
3. Save the file.

NOTE To use a file different from the *timesrvr.txt* file that contains the available time servers to use when creating a time stamp for a digital signature, change the setting of the ADSKSIGTIMESRVR environment variable. For example, change the variable to *c:\temp\mytimelist.txt*.

Quick Reference

Commands

SECURITYOPTIONS

Specifies password or digital signature options for a drawing file.

SIGVALIDATE

Displays information about the digital signature attached to a drawing file.

System Variables

SIGWARN

Controls whether a warning is presented when a file with an attached digital signature is opened.

Utilities

Attach Digital Signatures

Attaches a digital signature to files.

Digitally Sign an Encrypted Drawing

You can attach a password and a digital signature to a drawing file. The password must be attached first.

Modifications to files, including the adding of passwords, invalidate their digital signatures.

Quick Reference

Commands

SECURITYOPTIONS

Specifies password or digital signature options for a drawing file.

SIGVALIDATE

Displays information about the digital signature attached to a drawing file.

System Variables

SIGWARN

Controls whether a warning is presented when a file with an attached digital signature is opened.

Utilities

Attach Digital Signatures

Attaches a digital signature to files.

View Drawings with Digital Signatures

Digital signatures provide an electronic means of verifying the authenticity of signatures that are attached to drawing files.

Overview of Drawings with Digital Signatures

A digital signature identifies an individual or an organization through a digital ID (certificate), and enables you to validate the file. Validating a file is especially important when you are working on collaborative projects or transmitting files over the Internet. You can validate a digital signature in either the program or Windows Explorer.

Using the digital signatures feature, you can obtain the following information about signed files:

- Whether the file was changed since signed.
- Whether the signers are who they claim to be.
- Whether the signers can be traced (thus preventing forgery).

A digital signature is determined to be invalid for the following reasons:

- The file was corrupted when the digital signature was attached.
- The file was corrupted in transit.
- The digital ID is no longer valid.

NOTE If you want to retain valid digital signature status, do not add a password or otherwise modify or save a file that already has a digital signature attached. Make modifications, such as adding passwords, before signing the file. Signature information does not remain intact if you modify, save, or export drawing data.

View Digital Signature Details

When you receive a file that contains a digital signature, you can check whether the file was modified since it was signed, and other information.

An icon is displayed in the status bar when a file contains digital signature information. When you click the icon, you can verify the following information:

- Validity of the file
- Whether the file was modified since it was signed
- Individual or organization that signed the file
- Time stamp and comments attached to the file
- Issuer (certificate authority) of the digital ID
- "Valid to" and "valid from" dates for the digital ID

Digital signatures work on files that are compatible with AutoCAD 2000 and later drawing-file formats. If you receive a signed file and save the file, it is considered modified, and the signature is invalidated. However, you generally can still view information about it. If you receive a signed drawing-file and save the file, the signature information can be lost.

NOTE If the SIGWARN system variable is on and you open a file with a valid signature, the digital signature status is displayed. If SIGWARN is off and you open a file, the digital signature status is displayed only if a signature is invalid.

You can also view the properties of a drawing file in Windows Explorer to examine and validate a digital signature. Some operating systems display a modified icon for a drawing that has a valid digital signature attached. If the modified icon is displayed, your computer may slow down (depending on processor speed and file size), so you can turn off the display.

To validate a signature when opening an file

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Open and Save tab, select Display Digital Signature Information.
3. Click OK.
4. Open a signed file.

In the Digital Signature Contents dialog box, in the Other Fields list, select details of the digital signature, such as the issuer, beginning and expiration dates of the digital ID, and the serial number.

5 Click Close.

To validate a signature using the Validate Digital Signatures icon

1 Open a signed file.

2 On the status bar, click the Validate Digital Signatures icon.

3 In the Validate Digital Signatures dialog box, if the file has a valid digital signature, click View Base Signature.

The Digital Signature Contents dialog box displays details of the digital signature such as the signer, expiration date of the digital ID, serial number, comments about the digital signature, time stamp, and time service.

4 Click Close.

5 If the file contains an external reference (xref) that contains a digital signature, select View Xref Drawings in the Validate Digital Signatures dialog box. Click View Xref Signature to view the external references associated with the file.

6 Click Close.

To validate a signature in Windows Explorer

1 In Windows Explorer, locate a file whose digital signature you want to check.

Some operating systems display a modified icon for a drawing that has a valid digital signature attached.

2 Right-click the file name. Click Properties.

3 In the Properties dialog box, if the file contains a digital signature, the Digital Signature tab is displayed. Click the Digital Signatures tab to view the digital signature details.

A Valid Signature icon is displayed if a digital signature is valid, and an Invalid Signature icon is displayed if the digital signature is not valid.

4 In the Other Fields list, select an option from the list to view details of the digital signature, such as the issuer, description, and expiration date of the digital ID. You can also view the time stamp and time service.

5 When you have finished viewing the digital signature information, click OK.

To identify a signed file in Windows Explorer

- In Windows Explorer, locate files that you want to check for a digital signature. Signed files are displayed with the Digital Signatures icon.



To turn on or off the Digital Signatures icon display in Windows Explorer

1. In Windows Explorer, locate a file that has a digital signature icon.
2. Right-click the file name. Click Enable/Disable Digital Signature Icons.
3. Click the check box next to Validate Digital Signatures and Display Special Icons to turn on the icon display in Windows Explorer, or clear the check box to turn off the icon display.
4. Click OK. Log out of Windows, and then log back in.

To prevent signature status from being displayed for valid signatures

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Open and Save tab, clear the Display Digital Signature Information option.
3. Click OK.

Quick Reference

Commands

SECURITYOPTIONS

Specifies password or digital signature options for a drawing file.

SIGVALIDATE

Displays information about the digital signature attached to a drawing file.

System Variables

SIGWARN

Controls whether a warning is presented when a file with an attached digital signature is opened.

Utilities

Attach Digital Signatures

Attaches a digital signature to files.

Install a Root Certificate

When you receive a file that contains a digital signature from a certificate authority that is not recognized by your operating system, you may need to obtain the root certificate (the highest level of certificate in a certificate chain) before you can determine if the digital ID in the digital signature is valid.

Digital IDs work on the basis of a trust hierarchy. In a trust hierarchy, the root certificate is the digital ID of the issuing certificate authority. Popular browsers such as Microsoft Internet Explorer already include the root certificates of the major certificate vendors, making those digital IDs automatically trusted.

When you receive a file that is signed with a digital ID issued by someone other than a major vendor (for example, your company's internal IT department), you cannot validate the digital ID until you receive the root certificate.

To obtain a root certificate

1. To request the root certificate, contact the organization or individual who attached a digital signature to the current file.
2. When you receive the root certificate, click Start menu (Windows) ➤ Settings ➤ Control Panel ➤ Internet Options.
3. In the Internet Properties dialog box, Content tab, click Certificates.
4. In the Certificate Manager, click Import to run the Certificate Manager Import Wizard.
5. Follow the on-screen instructions to obtain a root certificate for the current file.

Quick Reference

Commands

SECURITYOPTIONS

Specifies password or digital signature options for a drawing file.

SIGVALIDATE

Displays information about the digital signature attached to a drawing file.

System Variables

SIGWARN

Controls whether a warning is presented when a file with an attached digital signature is opened.

Utilities

Attach Digital Signatures

Attaches a digital signature to files.

Use the Internet for Collaboration

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You can access and store drawings and related files on the Internet.

Get Started with Internet Access

Before you can transfer or save files to an Internet or an intranet location, you have to get access permissions and take security precautions.

In this topic and others, the term *Internet* is used to refer to both the Internet and an intranet. An intranet is a private network that uses the same standards as the Internet.

To save files to an Internet location, you must have sufficient access rights to the directory where the files are stored. Contact your network administrator or Internet service provider (ISP) to receive access rights for you and anyone else who needs to work with the files.

To use the Internet features with this program, you must have Internet or intranet access and Microsoft Internet Explorer 6.1 Service Pack 1 (or a later version).

If you connect to the Internet through your company's network, you might have to set up a proxy server configuration. Proxy servers act as security barriers by shielding information on your company's network from potential security risks due to external Internet access.

See the Internet applet in the Windows® Control Panel or contact your network administrator for details about how to configure a proxy server in your network environment.

Quick Reference

Commands

BROWSER

Launches the default web browser defined in your system's registry.

HYPERLINK

Attaches a hyperlink to an object or modifies an existing hyperlink.

System Variables

HYPERLINKBASE

Specifies the path used for all relative hyperlinks in the drawing.

INETLOCATION

Stores the Internet location used by the BROWSER command and the Browse the Web dialog box.

Add Hyperlinks to a Drawing

You can add hyperlinks that provide jumps in your drawings to specific files or websites.

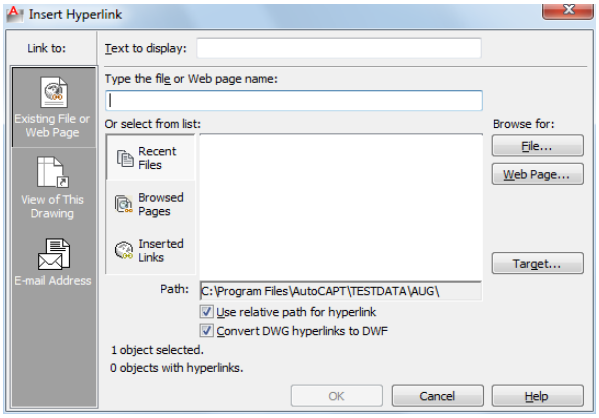
Overview of Using Hyperlinks in a Drawing

Hyperlinks provide a simple and powerful way to quickly associate a variety of documents (such as other drawings, bills of materials, or project schedules) with a drawing.

Hyperlinks are pointers you create in your drawings that provide jumps to associated files. For example, you can create a hyperlink that launches a word processing program and opens a specific file, or a hyperlink that activates your web browser and loads a particular HTML page. You can also specify a named location to jump to in a file, such as a view in a drawing file or a bookmark in a word processing program. You can attach hyperlinks to any graphical object in an AutoCAD drawing. Hyperlinks provide a simple and powerful way to quickly associate a variety of documents (such as other drawings, bills of materials, or project schedules) with an AutoCAD drawing.

You can create both *full* and *relative* hyperlinks in your drawings. Full hyperlinks store the fully specified path to a file location. Relative hyperlinks store a

partial path to a file location, relative to a default URL or directory you specify using the HYPERLINKBASE system variable.



Quick Reference

Commands

ATTACHURL

Attaches hyperlinks to objects or areas in a drawing.

DETACHURL

Removes hyperlinks in a drawing.

GOTOURL

Opens the file or web page associated with the hyperlink attached to an object.

HYPERLINK

Attaches a hyperlink to an object or modifies an existing hyperlink.

System Variables

HYPERLINKBASE

Specifies the path used for all relative hyperlinks in the drawing.

Use a Hyperlink to Access a File or a Web Page

Hyperlinks can point to files that are stored locally, on a network drive, or on the Internet, or to named locations in drawings, such as views.

By default, the hyperlink cursor and tooltip are displayed when the crosshairs are positioned over an object that has an attached hyperlink. You can then hold down the CTRL key and click (CTRL+click) to follow the link.

NOTE The PICKFIRST system variable must be set to 1 to open files associated with hyperlinks. You can turn off the hyperlink cursor, tooltips, and shortcut menu in the Options dialog box on the User Preferences tab.



When you create a hyperlink to a drawing and specify a named view to jump to, that view is restored when the hyperlink is opened. You can also create a hyperlink that opens a drawing in a specific layout.

Use Relative and Full (Absolute) Paths in Hyperlinks

Relative paths provide greater flexibility and are easier to edit than full hyperlinks. With relative hyperlinks, you can update the relative path for all the hyperlinks in your drawing at the same time, rather than editing each hyperlink individually.

You must use a full hyperlink when you create a link to a document that is not contained in the same file or whose relative path you don't know. For example, you would use a full hyperlink to link to a drawing on a different network drive. If you subsequently move the files referenced by full hyperlinks to a different directory, editing the hyperlink paths can be time consuming.

To create a full hyperlink to another file

1. In the drawing area, select one or more graphical objects to attach the hyperlink to.
2. Click Insert tab ➤ Data panel ➤ Hyperlink. 

3 Do one of the following:

- Under Type the File or Web Page Name, enter the path and name of the file that you want to associate with the hyperlink.
- Click the File button. Navigate to the location of the file that you want to associate with the hyperlink. Click Open.

4 (Optional) If you are creating a hyperlink to a drawing, select Target to specify a named location in the drawing to jump to and do the following:

- Select a named location to jump to.
- Click OK.

5 (Optional) Enter a description for the hyperlink in Text to Display.

6 Click OK.

To create a hyperlink to a named location

1 Create a hyperlink (see To create a full hyperlink to another file on page 2080).

2 Click Insert tab ➤ Data panel ➤ Hyperlink. 

3 Select the graphical object that the hyperlink is associated with.

4 In Type the File or Web Page Name, enter a pound sign (#) after the name of the file that the hyperlink is linked to, and then enter the named location. Do not include any spaces between the name of the file that the hyperlink is linked to, the pound sign (#), and the named location.

5 Click OK.

To edit a hyperlink attached to a graphical object

1 Click Insert tab ➤ Data panel ➤ Hyperlink. 

2 In the drawing area, select one or more graphical objects that use the same hyperlink.

3 Specify new values in the Edit Hyperlink dialog box. Click OK.

To remove a hyperlink from a graphical object

1. In the drawing area, select one or more graphical objects that use the same hyperlink.
2. Click Insert tab ➤ Data panel ➤ Hyperlink. 
3. In the drawing area, select one or more graphical objects that use the same hyperlink.
4. Click Remove Hyperlink. Click OK.

To open a file associated with a hyperlink

1. In the drawing area, select a graphical object with an attached hyperlink.
2. Use one of the following methods to follow the link:
   - Hold down the CTRL key and click.
   - Right-click. Click Hyperlink. Select the name of the hyperlink you want. Each hyperlink has either a description or the full URL to the referenced file.

NOTE If you have turned off display of the hyperlink cursor, Hyperlink options are not available on the shortcut menu. You can turn on the hyperlink cursor, tooltips, and shortcut menu in the Options dialog box on the User Preferences tab. The PICKFIRST system variable must be set to 1 to open files associated with hyperlinks.

To turn the hyperlink cursor display on or off

1. Click Application menu ➤ Options button.
2. In the Options dialog box, User Preferences tab, select or clear Display Hyperlink Cursor, Tooltip, and Shortcut Menu.
3. Click Apply or click OK.

To set the relative path for all hyperlinks in a drawing

1. Click Application menu ➤ Drawing Utilities ➤ Drawing Properties. 

2 On the Summary tab, enter a relative path in Hyperlink Base.

3 Click OK.

To create a relative hyperlink

1 Click Insert tab ➤ Data panel ➤ Hyperlink. 

2 In the drawing area, select one or more graphical objects to attach the hyperlink to.

3 Enter the name of the file to attach in Type the File or Web Page Name.

Do not enter any path information with the file name, or you will create a full hyperlink.

4 Click OK.

Quick Reference

Commands

ATTACHURL

Attaches hyperlinks to objects or areas in a drawing.

DETACHURL

Removes hyperlinks in a drawing.

DWGPROPS

Sets and displays the file properties of the current drawing.

GOTOURL

Opens the file or web page associated with the hyperlink attached to an object.

HYPERLINK

Attaches a hyperlink to an object or modifies an existing hyperlink.

HYPERLINKOPTIONS

Controls the display of the hyperlink cursor, tooltips, and shortcut menu.

OPTIONS

Customizes the program settings.

System Variables

HYPERLINKBASE

Specifies the path used for all relative hyperlinks in the drawing.

PICKFIRST

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

Use a Hyperlink to Create an Email Message

Typically you use hyperlinks in a drawing to associate graphical objects with related documents. In some cases, however, it might be useful to insert a hyperlink that creates an email message when the hyperlink is activated.

For example, consider a design project with several drafters responsible for updating different components of a drawing. As drafters add new components to the drawing, they can associate hyperlinks that include their email addresses with the components. This provides a convenient way to identify who created various drawing components and to send that person an email message with any design-related issues.

To create an email hyperlink

1. In the drawing area, select one or more graphical objects to attach the hyperlink to.
2. Click Blocks & References tab ➤ Data panel ➤ Hyperlink. 
3. In the Insert Hyperlink dialog box, under Link To, click E-mail Address.
4. Enter the recipient's address in E-mail Address.
5. (Optional) Specify a subject for the email message in Subject.
6. Click OK.

   When you open an email hyperlink, your default email application creates a new email message with the address and subject (if specified) filled out. You can then enter message text and send it by email.

Quick Reference

Commands

HYPERLINK

Attaches a hyperlink to an object or modifies an existing hyperlink.

System Variables

PICKFIRST

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

Use Hyperlinks in Blocks

Hyperlinks can be associated with blocks, including nested objects contained within blocks. If the blocks contain any relative hyperlinks, the relative hyperlinks adopt the relative base path of the current drawing when you insert them.

When you select a block element in the drawing area that contains multiple hyperlinks, the available hyperlinks are listed in the Hyperlink shortcut menu. You can activate any hyperlink associated with the currently selected block element, and edit or remove a hyperlink associated with a particular block instance.

For example, suppose you have a block that contains a line and a circle. The line has an attached hyperlink named *line*, and the circle has an attached hyperlink named *circle*. This particular instance of the block has a hyperlink named *block*, which you attached after the block was inserted. When you select an element in the block, you can open the hyperlink associated with that element, or any hyperlink associated with that particular instance of the block. If, for example, you select the line block element, you can open the *line* hyperlink or the *block* hyperlink. If you select the circle block element, you can open the *circle* hyperlink or the *block* hyperlink. In both cases, you can edit or detach only the *block* hyperlink associated with this instance of the block.

To edit or remove hyperlinks nested in a block, you must first explode the block or use the REFEDIT command.

To open a file associated with a hyperlink contained in a block

1. In the drawing area, select a block with attached hyperlinks.

2 Use one of the following methods to follow the link:

- Hold down the CTRL key and click.
- Right-click. Click Hyperlink. Select the name of the hyperlink you want. Each hyperlink has either a description or the full URL to the referenced file.

Quick Reference

Commands

ATTACHURL

Attaches hyperlinks to objects or areas in a drawing.

DETACHURL

Removes hyperlinks in a drawing.

GOTOURL

Opens the file or web page associated with the hyperlink attached to an object.

HYPERLINK

Attaches a hyperlink to an object or modifies an existing hyperlink.

Use a Hyperlink to Start a New Drawing

Hyperlinks can be associated with drawing template files to quickly start new drawings.

When you activate a hyperlink that points to a drawing template (DWT) file, a new drawing file is created based on the template rather than opening the actual template file.

This provides a convenient mechanism for creating a set of standardized drawing templates that you can share with others. Because the hyperlinked template files open a new drawing, there is no risk of others accidentally overwriting the original template.

To open a drawing template (DWT) file associated with a hyperlink

1 In the drawing area, select a graphical object with an attached hyperlink to a DWT file.

2 Use one of the following methods to follow the link:

- Hold down the CTRL key and click.
- Right-click. Click Hyperlink. Select the name of the hyperlink you want. Each hyperlink has either a description or the full URL to the referenced file.

A new drawing opens based on the hyperlinked DWT file.

NOTE If you have turned off display of the hyperlink cursor, Hyperlink options are not available on the shortcut menu. You can turn on the hyperlink cursor, tooltips, and shortcut menu in the Options dialog box on the User Preferences tab.

Quick Reference

Commands

GOTOURL

Opens the file or web page associated with the hyperlink attached to an object.

HYPERLINK

Attaches a hyperlink to an object or modifies an existing hyperlink.

System Variables

PICKFIRST

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

Work with Drawing Files over the Internet

You can open and save drawings to an Internet location, attach externally referenced drawings stored on the Internet, use i-drop to insert blocks by dragging drawings from a website, create a transmittal package of drawings that automatically includes all related files, and edit files online using the AutoCAD WS application.

Open and Save Drawing Files from the Internet

The file input and output commands recognize any valid Uniform Resource Locator (URL) path to a DWG file.

You can use AutoCAD to open and save files from the Internet. The AutoCAD file input and output commands (OPEN, EXPORT, and so on) recognize any valid URL path to an AutoCAD file. The drawing file that you specify is downloaded to your computer and opened in the AutoCAD drawing area. You can then edit the drawing and save it, either locally or back to any Internet or intranet location for which you have sufficient access privileges.

If you know the URL to the file you want to open, you can enter it directly in the Select File dialog box. You can also browse defined FTP sites or web folders in the Select File dialog box.

Use the Browse the Web dialog box to navigate to the Internet location where the file is stored, or access project collaboration sites hosted by Autodesk® Buzzsaw® using the Buzzsaw icon in the Select File or Save Drawing As dialog box.

Use the Browse the Web Dialog Box

With the Browse the Web dialog box, you can navigate quickly to a specific Internet location to open or save a file. You can specify a default Internet location that is loaded each time you open the Browse the Web dialog box. Using the Browse the Web dialog box to access files is particularly useful when you don't know the correct URL, or when you want to avoid entering a long URL each time you access an Internet location.

To open a drawing file from the Internet by entering a URL

1. Click File menu ➤ Open.
2. In the Select File dialog box, enter the URL to the file in File Name. Click Open.

   You must enter the Hypertext Transfer Protocol or File Transfer Protocol (for example, *http://* or *ftp://*) and the extension (for example, *.dwg* or *.dwt*) of the file you want to open.

To open a drawing file from the Internet by browsing an FTP site

1. Click File menu ➤ Open.
2. In the Select File dialog box, click Tools ➤ Add/Modify FTP Locations.

3 In the Add/Modify FTP Locations dialog box, under Name of FTP Site, enter the site name for the FTP location (for example, **ftp.autodesk.com**).

4 Under Log On As, select one of the following:

- **Anonymous.** Logs on to the FTP site as an anonymous user. If the FTP site does not allow anonymous logons, select User and enter a valid user name.
- **User.** Logs on to the FTP site using the specified user name.

5 Enter a password if required by the FTP site.

6 Click Add. Click OK.

7 In the Select File dialog box, select FTP from the Places list.

8 Double-click one of the FTP sites and select a file.

9 Click Open.

To open a drawing file from the Internet by browsing a web folder

1 Click File menu ➤ Open.

2 In the Select File dialog box, under Look In, select Web Folders (or My Network Places, depending on your version of Microsoft® Windows®).

3 Double-click one of the web folders and select a file.

You can create new web folders in Windows Explorer. Consult your system administrator or Windows Explorer Help for more information about web folders.

4 Click Open.

To save a drawing file to the Internet by entering a URL

1 Click File menu ➤ Save As.

2 Enter the URL to the file in File Name.

You must enter the File Transfer Protocol or Hypertext Transfer Protocol (for example, *ftp://* or *http://*) and the extension (for example, *.dwg* or *.dwt*) of the file you want to save. You must have access rights to save files to the specified location.

3 Select a file format from the Files of Type list. Click Save.

To save a drawing file to the Internet by browsing an FTP site

1. Click File menu ➤ Save As.
2. In the Save Drawing As dialog box, select FTP from the Places list.

   If you have not yet added the FTP site to the available FTP locations, click Tools ➤ Add/Modify FTP Locations and define the FTP location.
3. Double-click one of the FTP sites, and select a file.
4. Select a file format from the Files of Type list. Click Save.

To save an file to the Internet by browsing a web folder

1. Click File menu ➤ Save As.
2. In the Select File dialog box, under Look In, select Web Folders (or My Network Places, depending on your version of Windows).
3. Double-click one of the web folders.

   You can create new web folders in Windows Explorer. Consult your system administrator or Windows Explorer Help for more information about web folders.
4. Specify a file name and select a file format from the Files of Type list.
5. Click Save.

   You must have access rights to save files to the specified location.

To specify the default Internet location used by the Browse the Web dialog box

1. Click Tools menu ➤ Options.
2. On the Files tab, click the plus sign next to Help and Miscellaneous File Names.
3. Select the Default Internet Location option. Click Browse.
4. In the Select a File dialog box, do one of the following:
   - Enter the URL to an Internet location. Click Open.
   - Click the Search the Web button and enter the Internet address in the Look In box.
5. Click OK.

To open an file from the Internet with the Browse the Web dialog box

1. Click File menu ➤ Open.
2. Click the Search the Web button.

   Many Internet connections require you to enter a valid user name and password before accessing the Internet. AutoCAD prompts you to enter this information.
3. In the Browse the Web dialog box, do one of the following:
   - Click a hyperlink on the HTML page that is displayed.
   - Enter a full or partial URL in Look In, and then press ENTER.
4. Repeat step 3 until you locate the file that you want to open.
5. Click Open.

To save an file to an Internet location with the Browse the Web dialog box

1. Click File menu ➤ Save As.
2. Click the Search the Web button.

   Many Internet connections require you to enter a valid user name and password before accessing the Internet. AutoCAD prompts you to enter this information.
3. In the Browse the Web dialog box, do one of the following:
   - Click a hyperlink on the HTML page that is displayed.
   - Enter a full or partial URL in Look In, and then press ENTER.
4. Repeat step 3 until you navigate to the location where you want to save the file.
5. Click Save.

Quick Reference

Commands

APPLOAD

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

EXPORT

Saves the objects in a drawing to a different file format.

HYPERLINK

Attaches a hyperlink to an object or modifies an existing hyperlink.

OPEN

Opens an existing drawing file.

Share Drawing Files Internationally

Beginning with AutoCAD 2007-based products, drawing files and most files associated with drawing files use the Unicode standard. This lets you maintain both the visual fidelity and data integrity of international characters when you save and open drawing files.

NOTE AutoCAD 2006, AutoCAD LT 2006, and prior versions were not Unicode applications. When sharing drawings with earlier, non-Unicode, versions, use ASCII characters to ensure compatibility when you save files, insert xrefs, and specify folder paths.

Overview of Unicode

All characters are processed numerically by the computer operating system, which assigns a number to each character. Various numeric encoding systems have been used in the past, however these encoding systems often conflicted. As a result, operating systems and applications relied on *code pages* with specific character sets and numbering assigned to countries or regions.

To facilitate international compatibility, the Unicode standard was adopted by major industry leaders and is being maintained by the Unicode Consortium.

Drawing File Impact

Language-specific characters can be used in file names and text within drawing files, or files associated with drawing files. The following are common examples:

- Drawing file names
- Folder path names
- Sheet set data files
- Named objects such as layers and blocks within a drawing

- Linetype and hatch pattern file names and their contents
- Text used in notes and dimensions within a drawing
- DWF or DWFx markup files

This means that drawings can be opened, worked on, and saved worldwide regardless of language-specific characters. The only requirement is that the appropriate language pack must be installed first.

When you save text files such as linetype (LIN), hatch pattern (PAT), and script (SCR) files using an application such as Notepad, it is recommended that you specify Unicode encoding at the bottom of the Save As dialog box to ensure compatibility.

Limitations

Most international drawing projects can be completed within the current product environment. However, there are several file types and features that are not supported yet between countries and regions that use different Windows code pages. These features include the following:

- eTransmit
- Round trip file and data compatibility with non-Unicode products
- Block attributes
- Visual LISP files

NOTE You can still use eTransmit to convey transmittal sets to countries and regions that use the same code page.

Quick Reference

Commands

OPEN

Opens an existing drawing file.

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.

System Variables

TEXTOUTPUTFILEFORMAT

Provides Unicode options for log files.

Access Buzzsaw for Project Collaboration

Using Autodesk® Buzzsaw® you can store, manage, and share documents that populate a Buzzsaw site.

Buzzsaw is a secure, online project collaboration service in which members in different locations can post files to and access files from a centralized site. You can save files, send transmittal sets, and publish sheets to Buzzsaw.

You can get a free 30-day trial subscription when you follow the procedures to access Buzzsaw, or from the Autodesk website. Your Buzzsaw subscription includes user licenses to access Buzzsaw. When you invite your consultants and contractors to your site, the Buzzsaw software is automatically downloaded.

Prepare to Use Buzzsaw

To use Buzzsaw, you must already have a project hosting account or be given access to a subscriber's Buzzsaw site. The subscriber will provide you with the Buzzsaw URL, user name, and password so that you can log in, access, and post files.

Use Buzzsaw

You can access Buzzsaw from several standard file selection dialog boxes (such as New or Open). The Buzzsaw icon is displayed in the Places list for quick access.

- **Sheet Set Manager Support.** Stores information necessary to publish sheet sets and transmittal sets. For more information about the Sheet Set Manager, see Work with Sheets in a Sheet Set on page 471.
- **Publish Support.** Publish DWF or DWFx files directly to Autodesk Buzzsaw locations. For more information about publishing to an Autodesk Buzzsaw location, see Set Publish Options on page 1753.

- **eTransmit Support.** You can send a transmittal set of DWG files to a Buzzsaw site and you can create and send a transmittal set using the Sheet Set Manager.

If you plan to use Buzzsaw for project collaboration, set up some shortcuts so you can navigate to frequently used sites more quickly. Using the Buzzsaw icon in the Places list, you can specify a shortcut to an existing project collaboration site from the list or create a new shortcut using the Add a Buzzsaw Location Shortcut.

You can right-click in the Location area of a dialog box that supports Buzzsaw to display a shortcut menu that provides options to add, manage, or delete Autodesk Buzzsaw locations. Changes you make to locations get stored in the registry.

To access a Buzzsaw site for the first time

1. In a file navigation dialog box, such as the Select File dialog box, in the Places list, click the Buzzsaw icon.

   NOTE The first time you select Buzzsaw, Internet Explorer is launched and the Autodesk Buzzsaw web page is displayed.

   Buzzsaw Location Shortcuts is displayed in the Look In box.

2. Double-click Add a Buzzsaw Location Shortcut.
3. In the Log In To Buzzsaw Site dialog box, under Buzzsaw Site, enter the location name for the Buzzsaw site you want to access.
4. Under Login, Buzzsaw User Name, enter a login name.

   This name is provided by the person granting you access to the Buzzsaw site.

5. In the Password box, enter a password. Click OK.

   AutoCAD verifies that you have access to the site specified. If access is granted, the Create a Buzzsaw Location Shortcut is displayed. If access is not granted, an error message is displayed.

   NOTE Select the Save Login Name and Password check box, if you want to save your login name and password for future use.

6. In the Create a Buzzsaw Location Shortcut dialog box, under Select a Buzzsaw Folder, Project, or Document Set, click Browse.

7 In the Select a Buzzsaw Location dialog box, navigate to a project or folder you want to access. Click OK.

The path is displayed in the Select a Buzzsaw Folder, Project, or Document Set box.

8 Under Enter a Name for This Buzzsaw Location Shortcut, enter a name for the shortcut. Click OK.

You are returned to the File navigation dialog box; the shortcut has been created. You can double-click the shortcut to access the Buzzsaw location specified by the shortcut. You can create as many Buzzsaw shortcuts as needed.

To access a Buzzsaw site

1 In a file navigation dialog box that supports Buzzsaw, in the Places list, click the Buzzsaw icon.

Buzzsaw Location Shortcuts is displayed in the Look In box along with any configured Buzzsaw shortcuts.

2 Double-click a Buzzsaw shortcut.

If you have not logged in to the site, the Log In To Buzzsaw dialog box is displayed. After logging in, you go to the location defined by that shortcut.

To add a Buzzsaw location shortcut

1 In a file navigation dialog box that supports Buzzsaw, in the Places list, click the Buzzsaw icon.

Buzzsaw Location Shortcuts is displayed in the Look In box along with any configured Buzzsaw shortcuts.

2 Right-click a Buzzsaw shortcut. Click Add a Buzzsaw Location Shortcut.

3 In the Log In To Buzzsaw Site dialog box, under Buzzsaw Site, select the location for the Buzzsaw site to access.

4 Under Login, Buzzsaw User Name, enter a login name, if necessary.

(The user name is provided to you by the person granting you access to the Buzzsaw site.)

5 Under Login, Password, enter a password. Click OK.

NOTE Check the box, Save Login Name and Password, if you want to save your login name and password on this computer for future use.

6. In the Create a Buzzsaw Location Shortcut dialog box, under Select a Buzzsaw Folder, Project, or Document Set, click the Browse button to navigate to the location of the project or folder.
7. In the directory tree of the Select a Buzzsaw Location dialog box, click the site name to view the contents and navigate to either a project or folder. Click OK.

   The path is now displayed in the Create a Buzzsaw Location Shortcut dialog box, under Select a Buzzsaw Folder, Project, or Document Set.
8. Under Enter a Name for This Buzzsaw Location Shortcut, enter a name. Click OK.

   This folder name is added to the Site list in the navigation dialog box.

To rename a Buzzsaw location shortcut

1. In a file navigation dialog box that supports Buzzsaw, in the Places list, click the Buzzsaw icon.

   Buzzsaw Location Shortcuts is displayed in the Look In box along with any configured Buzzsaw shortcuts.
2. Right-click a Buzzsaw shortcut. Click Manage Buzzsaw Location Shortcuts.
3. In the Buzzsaw Location Shortcuts dialog box, select a shortcut. Click Rename.
4. Enter a new name for the Buzzsaw location shortcut. Click Close.

To modify a Buzzsaw location shortcut

1. In a file navigation dialog box that supports Buzzsaw, in the Places list, click the Buzzsaw icon.

   Buzzsaw Location Shortcuts is displayed in the Look In box along with any configured Buzzsaw shortcuts.
2. Right-click a Buzzsaw shortcut. Click Manage Buzzsaw Location Shortcuts.
3. In the Buzzsaw Location Shortcuts dialog box, select a shortcut. Click Modify.
4. In the Log In To Buzzsaw Site dialog box, click to log in to the Buzzsaw site.
5. In the Edit a Buzzsaw Location Shortcut dialog box, modify the shortcut location or the shortcut name.

6 Click OK. Click Close.

To delete a Buzzsaw location shortcut

1 In a file navigation dialog box that supports Buzzsaw, in the Places list, click the Buzzsaw icon.

Buzzsaw Location Shortcuts is displayed in the Look In box along with any configured Buzzsaw shortcuts.

2 Right-click a Buzzsaw shortcut. Click Manage Buzzsaw Location Shortcuts.

3 In the Buzzsaw Location Shortcuts dialog box, select a shortcut. Click Delete.

The selected shortcut is deleted.

4 Click Close.

To save a file opened in a Buzzsaw site to a different Buzzsaw site

1 Click File menu ➤ Save.

2 In the Save Drawing As dialog box, under File Name, the entire path to the drawing file is displayed. Click Save.

To save a file opened outside of Buzzsaw to a Buzzsaw site

1 Click File menu ➤ Save As.

2 In the Save Drawing As dialog box, in the Places list, click the Buzzsaw icon.

3 Double-click an existing shortcut or create a new shortcut.

NOTE If you have not logged into the site, the Log In To Buzzsaw dialog box is displayed. After logging in, you go to the location defined by that shortcut.

4 Enter a file name and click Save.

To send a transmittal set to Buzzsaw

1 Click Output tab ➤ Send panel ➤ eTransmit. 

2 In the Create Transmittal dialog box, click OK.

3 In the Specify Zip File dialog box, in the Places list, click the Buzzsaw icon.

Buzzsaw Location Shortcuts is displayed in the Save In box. Existing Buzzsaw shortcuts are displayed in the Site column.

4 Double-click an existing shortcut or create a new shortcut.

NOTE If you have not logged into the site, the Log In To Buzzsaw dialog box is displayed. After logging in, you go to the location defined by that shortcut.

5 Click Save.

The transmittal package is created.

To send a transmittal set to Buzzsaw from the Sheet Set Manager

1 In the Sheet Set Manager, Sheet List tab, right-click. Click ETransmit.

The program gathers the information to create the transmittal package.

2 In the Create Transmittal dialog box, click OK.

3 In the Specify Zip File dialog box, in the Places list, click the Buzzsaw icon.

Buzzsaw Location Shortcuts is displayed in the Look In box along with any configured Buzzsaw shortcuts.

4 Double-click an existing shortcut or create a new shortcut.

NOTE If you have not logged in to the site, the Log In To Buzzsaw dialog box is displayed. After logging in, you go to the location defined by that shortcut.

5 Click Save.

The transmittal package is created.

To publish a file to Buzzsaw

1 In a drawing, click File menu ➤ Publish.

2 In the Publish dialog box, under Publish To, select one of the following options and click Publish Options.

- Plotter Named in Page Setup
- DWF Format (DWF File or DWFx File)

3 In the Publish Options dialog box, under Default Output Location, click Location. Click the [...] button.

4 In the Select a Folder for Generated Files dialog box, in the Places list, click the Buzzsaw icon.

Buzzsaw Location Shortcuts is displayed in the Look In box along with any configured Buzzsaw shortcuts.

5 Double-click an existing shortcut or create a new shortcut.

NOTE If you have not logged into the site, the Log In To Buzzsaw dialog box is displayed. After logging in, you go to the location defined by that shortcut.

6 Click Select.

7 In the Publish Options dialog box, click OK

8 In the Publish dialog box, click Publish.

9 In the Specify DWF File dialog box, click Select.

Publishing begins. A status box indicates when publishing is complete.

To publish a sheet set to Buzzsaw

1 In the Sheet Set Manager, Sheet List tab, click the arrow next to Publish. Select Publish Dialog Box.

2 In the Publish dialog box, click Sheet Set Publish Options.

3 In the Sheet Set Publish Options dialog box, under Default Output Location, click Location. Click the [...] button.

4 In the Select a Folder for Generated Files dialog box, in the Places list, click the Buzzsaw icon.

Buzzsaw Location Shortcuts is displayed in the Look In box along with any configured Buzzsaw shortcuts.

5 Double-click an existing shortcut or create a new shortcut.

NOTE If you have not logged into the site, the Log In To Buzzsaw dialog box is displayed. After logging in, you go to the location defined by that shortcut.

6 Click Select.

7 In the Sheet Set Publish Options dialog box, click OK.

8 In the Publish dialog box, click Publish.

9 In the Specify DWF File dialog box, click Select.
Publishing begins. A status box indicates when publishing is complete.

Quick Reference

Commands

OPEN

Opens an existing drawing file.

Use AutoCAD WS for Drawing File Collaboration

Use AutoCAD® WS to share, edit, and manage AutoCAD drawings on the Web.

AutoCAD WS is an application that interfaces directly with AutoCAD. Changes to your local AutoCAD drawings are synchronized with the online copies that you have stored on the AutoCAD WS server.

The AutoCAD WS Editor allows you to access and edit the online copies from any computer with a web browser. Multiple users can work on the same drawing file online and in real time.

Access and Share Drawings in AutoCAD WS

Log in to your AutoCAD WS account to access and manage your uploaded files from any computer with a web browser and Internet access.

Share your drawings and other files online with other users. Users can view, edit, or download a shared drawing without having to install CAD software or a DWG viewer. They can also download other shared files, such as PDFs, ZIPs, raster images, and any other files.

Collaborate in Real Time

After sharing your online drawing, collaborate with other users in real time by inviting them to simultaneously view and edit the drawing.

Use the Timeline

The timeline allows you to track the history of an online drawing. Use the timeline to view previous versions of the drawing, including versions resulting

from real-time collaborations. All comments made over the progress of a drawing are displayed.

While viewing a past version of the drawing, you cannot make changes. Save a copy of the drawing to continue working on the selected version.

For additional information and tutorials on how to use AutoCAD WS, see the *AutoCAD WS* website.

To auto-upload drawings and files

- Click Online tab ➤ Upload panel ➤ Upload. 
  The current drawing and any related external references (xrefs, images, or underlays), fonts, and stylesheet files are uploaded to your AutoCAD WS web storage.

  By default, these files are automatically uploaded every time the drawing is saved.

To upload selected files

1. Click Online tab ➤ Upload panel ➤ Upload Multiple Files. 
2. In the Choose Files to Upload dialog box, select the files to upload.

   You can upload files of any type, such as AutoCAD drawings (DWG OR DXF), PDFs, or Microsoft Office files.
3. Click Open.

To manage uploads

1. Click Online tab ➤ Upload panel ➤ Manage Uploads. 
2. In the Manage Uploads dialog box, do the following.
   - Select a drawing's check box to upload the latest version of the drawing and enable automatic uploads. The drawing is uploaded each time it is saved.
   - Clear a drawing's check box to cancel an upload if it is in progress, and disable automatic uploads.

NOTE You can stop and restart the uploading process for drawing files only.

3 To remove a file from the list, right-click the file and click Remove from This List.

To open and edit drawings online

- Click Online tab ➤ Content panel ➤ Open Online. 
  The current drawing is uploaded and opened in AutoCAD WS. Use the available online tools to edit the drawing and collaborate with others.

To view your online files in AutoCAD WS

1 Click Online tab ➤ Content panel ➤ Online Drawings. 
All drawings and files are displayed in the Drawings section in AutoCAD WS.

2 Double-click a drawing file to open it in the AutoCAD WS Editor.

NOTE Only drawing files can be opened in AutoCAD WS. To open other files, you must download them.

To select a previous version of the current drawing

1 Click Online tab ➤ Content panel ➤ Timeline. 
The Timeline dialog box displays all previous versions of the online copy of the current drawing.

2 Locate the desired version and click one of the following:
   - **View Online.** Opens the selected drawing version in AutoCAD WS.
   - **Download.** Saves a copy of the drawing version to your local drive.

To share the online copy of the current drawing in AutoCAD WS

1 Click Online tab ➤ Share panel ➤ Share Drawing. 

2 In the Share Drawing dialog box, enter the email addresses of the recipients, or select them from your AutoCAD WS contacts.

3 Define the access permissions for each recipient.

4 (Optional) Enter a message to include in the email notification.

5 Click Share.

To get a URL to the online copy of the current drawing

1 Click Online tab ➤ Share panel ➤ Get Link. 

2 In the Get Link dialog box, select the Share Link check box.

3 Define the access permissions for each recipient.

4 Click Copy to copy the link to your clipboard.

5 Click OK.

6 Paste the link into any text editor and share it with others.

To review messages

- Click Online tab ➤ Share panel ➤ Messages. 
  The AutoCAD WS Messages section is displayed.

Quick Reference

Commands

GETLINK

Generates a URL to the AutoCAD WS online copy of the current drawing.

MANAGEUPLOADS

Manages the uploading of files stored on the AutoCAD WS server.

MESSAGES

Displays messages from your AutoCAD WS account.

ONLINEDRAWINGS

Opens AutoCAD WS in a web browser and lists the available online files.

OPENONLINE

Opens the AutoCAD WS Editor in a web browser and displays an online copy of the current drawing for online editing and sharing.

SHARE

Shares the AutoCAD WS online copy of the current drawing with other users.

TIMELINE

Provides access to previous versions of the AutoCAD WS online copy of the current drawing.

UPLOAD

Uploads the current drawing to AutoCAD WS and controls whether to automatically upload changes.

UPLOADFILES

Uploads selected files to AutoCAD WS.

Work with Xrefs over the Internet

You can attach externally referenced drawings stored on the Internet or an intranet to drawings stored locally on your system.

For example, you might have a set of construction drawings that are modified daily by a number of contractors. These drawings are stored in a project directory on the Internet. You can maintain a master drawing on your computer, and attach the Internet drawings to the master drawing as external references (xrefs). When any of the Internet drawings are modified, the changes are included in your master drawing the next time you open it. This is a powerful mechanism for developing accurate, up-to-date composite drawings that can be shared by a design team.

NOTE If you have a slow Internet connection or are working with a master drawing that has many xrefs attached, the download of the xrefs to your system might take a long time.

To attach an xref to a drawing stored on the Internet

1. Click Insert menu ➤ External References.
2. In the Select Reference File dialog box, do one of the following:
   - Under File Name, enter the URL to the file that you want to attach. You must include the transfer protocol (for example, *http://* or *ftp://*) in the URL.
   - Click the Search the Web button, and then use the Browse the Web dialog box to navigate to the file you want to attach.

If your network or ISP requires a user name and password, you are prompted to enter this information.

3. Click Open.
4. In the External Reference dialog box, click OK.
5. Specify an insertion point for the xref.

Quick Reference

Commands

HYPERLINK

Attaches a hyperlink to an object or modifies an existing hyperlink.

Insert Content from a Website

With the Autodesk® i-drop® functionality, you can use a drag-and-drop method to insert web content into your drawing.

Overview of i-drop

i-drop is a drag-and-drop method of inserting content from the Web into your current drawing. Using i-drop, you can easily insert content such as drawings and catalog items into an open drawing. Specific examples of i-drop content that you can insert include a block of a chair or window or a bitmap of a linoleum sample.

You can use the Publish to Web wizard to create a web page that has i-drop-enabled content for a drawing file. For details about other ways to create a web page that has i-drop content, see the information about i-drop on the Autodesk website at the following URL:

http://www.autodesk.com/idrop

Quick Reference

Commands

SETIDROPHANDLER

Specifies the default type of i-drop content for the current Autodesk application.

Use i-drop

To use i-drop, you simply drag a representative i-drop content image from a web page into your drawing area. You can also specify the type of i-drop content for your application.

The representative i-drop content image will usually be a visual representation of what will appear in your drawing.

i-drop also offers the following functionality:

- **Set the default type of i-drop content accepted.** You can set the default type of i-drop content accepted by the program. The setting you select determines the type of content that is inserted in your drawing when you drag a representative i-drop content image from the Web. For example, if you select Block, the inserted content is a block.
- **Use right-click and drag to select the type of i-drop content.** You can select the type of i-drop content to insert in your drawing when you right-click and drag a representative i-drop content image.
- **Use right-click and drag to specify associated data files to download.** You can specify which associated data files to download when using right-click to drag i-drop content into a drawing.
- **Specify the location of downloaded associated data files.** You can specify where you want to download the data files that are associated with i-drop content.
- **View a log file of i-drop content.** You can view a log file of i-drop content for each drawing. This log file is saved in the same folder as the drawing.

To insert i-drop content into a drawing

1. Open the drawing in which you want to insert i-drop content.

2. Drag a representative i-drop content image from a web page into your current drawing.

   When you move your cursor over a representative i-drop content image on a website, the cursor changes to a dropper image.You are now in Insertion mode.

3. In your current drawing, click where you want the i-drop content inserted.

   The i-drop content is inserted into your drawing.

To specify options for data files associated with inserted i-drop content

You can specify the transfer and location of associated data files when you insert i-drop content into a drawing.

1. Right-click a representative i-drop content image and drag it from a web page into your current drawing.

   When you move your cursor over a representative i-drop content image on a website, the cursor changes to a dropper image.

2. On the shortcut menu, click Insert As ➤ Block.
3. In the i-drop Options Dialog box, Choose Associated File area, select the associated data files you want to download with your i-drop content.
4. In the Choose Destination Path area, the default destination path is displayed. You can enter a different path or click the [...] button to locate a different folder.
5. Click OK.
6. In the drawing, click where you want your content inserted.

To specify the default type of i-drop content

1. At the Command prompt, enter **setidrophandler**.
2. In the Set Default i-drop Content Type dialog box, in the Choose Default i-drop Content Type list, select a type of i-drop content.

   The only i-drop content types that are listed are those for installed Autodesk products that provide this type of i-drop functionality.

   If the representative i-drop content image you drag into your drawing points to more than one type of content, this setting specifies which type of content is actually inserted. Selecting Block specifies that a block is inserted. You can drop a block into a drawing that was created in any AutoCAD-based product.

To view a log file of i-drop content in your drawing

- In the folder where your drawing is stored, double-click the file named *<drawing name>_idrop.txt*.
  The log file displays the name of the drawing file (target file), the name of the i-drop content file inserted in your drawing (located at the end of the source URL), the date and time it was inserted, the source URL of each associated data file, and the destination (location) of any associated data files.

To view the URL hyperlink for the source of an inserted i-drop content image

1. In your drawing, select the i-drop content image.
2. Move your cursor over the selected i-drop content image.
   The URL hyperlink is displayed in a tooltip.

Quick Reference

Commands

SETIDROPHANDLER

Specifies the default type of i-drop content for the current Autodesk application.

Package a Set of Files for Internet Transmission

With eTransmit, you can package a set of files for Internet transmission. Drawing files in the transmittal package automatically include all related dependent files such as xrefs and font files.

A common problem when sending drawing files to someone is neglecting to include related dependent files such as xrefs and text fonts. In some cases, not including these files can make the drawing files unusable by the recipient. With eTransmit, dependent files are included automatically in the transmittal package, reducing the possibility of error.

Choose the Files to Package

The Create Transmittal dialog box displays two tabs for individual drawing files, or three tabs for sheet sets and drawing files that are referenced by sheet

sets. Using these tabs, you can view and change the files to be included in the transmittal package.

- **Sheets.** This tab is available only when a sheet set is open. It displays a hierarchical list of sheets in the current sheet set. When you check a sheet, all dependent files are automatically included in the transmittal package.
- **Files Tree.** This tab displays a list of files. You can expand or collapse each drawing file in the list to display its dependent files. By default, dependent files are automatically included in the transmittal package unless you uncheck them.
- **Files Table.** This tab displays a table of files, their folder locations, and file details. Each file can be checked or unchecked, providing the most direct control over the contents of the transmittal package. Files are not automatically checked or unchecked.

The following table lists the common file types that are automatically added to a transmittal package. You can choose to remove any of the files automatically added and you can manually add additional files that you want included in the transmittal package.

File Types Automatically Added to a Transmittal Package

File Type	Description
*.*dwg*	Root drawing file and any attached external references
Various image file types	Raster image files that are attached externally to the root drawing or external referenced drawings, or as part of a material
*.*dgn*	DGN files that are attached externally to the root drawing or external referenced drawings
*.*dwf*	Design Web Format files that are attached externally to the root drawing or external referenced drawings
*.*dwfx*	DWF file with XPS information that is attached externally to the root drawing or an externally referenced drawing

File Types Automatically Added to a Transmittal Package

File Type	Description
**.pdf*	PDF files that are attached externally to the root drawing or external referenced drawings
**.dxe*	Data extraction files that contain extraction settings and data link information
**.xls*	Microsoft Excel spreadsheet files that are linked to data extraction tables
**.fmp*	Font Mapping File used for the In-Place Text Editor when working with Multiline Text objects
**.ctb*	Color-dependent plot style files used to control the appearance of the objects in the drawings of the transmittal set when plotting
**.stb*	Named plot style files used to control the appearance of the objects in the drawings of the transmittal set when plotting
**.pc3*	Plotter configuration files used to control the device and settings for the device when plotting the layouts of the drawings in the transmittal set
**.dwt*	Drawing template file that is associated with a sheet set
**.dst*	Sheet set file
**.dws*	Drawing standards file that is associated to a drawing for standards checking

The following table lists the common file types that are not automatically added to a transmittal package. Many of these files include information that are specific to applications that run with the product, or contains information

that were originally used to format objects in the drawing and are not longer needed to properly view the drawing when sent to another user.

File Types Not Automatically Added to a Transmittal Package

File Type	Description
*.*arx*, *.*dbx*, *.*lsp*, *.*vlx*, *.*dvb*, *.*dll*	Custom application files (ObjectARX, ObjectDBX, AutoLISP, Visual LISP, VBA and .NET)
*.*shx*	Compiled shape files used for text styles or complex linetypes
*.*ttf*	True Type font files used for text styles
*.*pfa*, *.*pfb*	PostScript Type 1 font files
*.*pat*	Hatch pattern files
*.*lin*	Linetype definition files
*.*mln*	Multiline definition files
*.*pmp*	Plotter model parameter files, used to store configuration information specific to a plotter such as custom paper sizes
*.*unt*	Unit conversion file, used by QuickCalc and the CVUNIT AutoLISP function
*.*pss*	Plot stamp settings files
OLE objects (linked files)	OLE objects that are linked to files in the drawing are not included in the transmittal package
Rtext objects (linked files)	Text files that are referenced by the RTEXT object are not included in the transmittal package
*.*xls (Microsoft Excel files)*	Microsoft Excel spreadsheet files that are linked to data extraction tables are not included in the transmittal package

Include Instructions to the Recipient

A report file is automatically generated that includes a list of files in the transmittal package. The report also includes instructions describing what must be done with drawing-dependent files such as xrefs and font files so that they are usable with the included drawing files. You can also add your own notes to the report file.

Save Transmittal Setups

You will probably send transmittal packages multiple times during a project. The eTransmit feature provides a method to name and save your transmittal settings as *transmittal setups*. The Transmittal Setups dialog box displays a list of saved transmittal setups from which you can select each time you transmit a set of files. The default transmittal setup is named Standard.

Choose Transmittal Options

Several options are available for the transmittal package using transmittal setups. With these options, you can

- Package the transmittal package in a ZIP file, in a self-extracting EXE file, or in a folder that is copied to a specified location.
- Specify that the folder structure of the transmitted files be organized in a logical hierarchy, flattened into a single folder, or copied "as is" to the recipient's computer. If you specify an FTP or HTTP destination, the transmittal package uses the single folder option.
- Add password protection to the transmittal package, bind xrefs automatically, set the default plotter to "none," and other options.

After you create the transmittal package, you can post it to an Internet location or send it to others as an email attachment. If you want to send the transmittal package in an email, you can use an option in the Modify Transmittal Setup dialog box that automatically launches your default system email application. When the transmittal package is created, the transmittal package and transmittal report file are automatically attached to a new email.

NOTE Regardless of which folder structure option you select for the transmittal package, any fully specified (absolute) paths of dependent files are converted to relative paths or "no path" to ensure that the dependent files can be located by the drawing file.

Transmit Sheet Sets

In the Sheet Set Manager, you can create a transmittal package conveniently from a sheet set, a sheet subset, or a sheet. Use the Sheet Set Manager to specify what you want to transmit.

NOTE In the Sheet Set Manager, on the Sheet List tab, right-click a sheet set node, a sheet subset node, or a sheet node on the sheet tree. On the shortcut menu, the eTransmit option will assemble the correct sheets depending on the node that you selected. Use named sheet selections to restore large or complex selections of sheets.

The tabs in the Create Transmittal dialog box control which files are included in the transmittal package.

- On the Sheet List tab, you can easily control which of the listed sheets are to be transmitted by selecting or clearing the boxes next to the sheet names.
- On the Files Tree tab and the Files Table tab, you can add or remove drawing files and dependent files that are listed. You can also add or remove any other files.

NOTE If several sheets come from the same drawing file, only the first sheet associated with the drawing can be selected with a check mark or cleared of its selection on the Sheet List tab. The other sheets will not be available. For this reason, it is recommended that you create only one layout tab in each drawing file.

How Transmittal Packages Work with AutoCAD LT or Previous Versions of AutoCAD

When transmittal packages are sent between different people and organizations, it is possible that AutoCAD LT or previous versions of AutoCAD are involved in the transmittals.

You can transmit packages between users of different AutoCAD-based products, and you can use eTransmit to convert files to several previous DWG file formats. In addition, transmittal packages created in AutoCAD 2005 and later can contain sheet sets, which are not available in AutoCAD LT and earlier versions of AutoCAD. The files in these sheet sets can still be sent and received by AutoCAD LT and previous versions of AutoCAD, however, if you want a more powerful, convenient method of transmitting files, the best solution is to use sheet sets in transmittal packages.

When you convert drawings to a legacy drawing file format (AutoCAD 2007 or earlier), drawings containing objects that exceed the legacy large object size limits are reported as errors in the Conversion Error Encountered dialog box and are not converted.

Transmit Files Internationally

Beginning with AutoCAD 2007, the names of the files generated by AutoCAD-based products use Unicode standards and can be shared internationally. However, eTransmit uses WinZip technology to create transmittal packages. Because WinZip currently is not a Unicode-compliant application, it is recommended that you use one of the following alternatives:

- Transmit files individually.
- Use eTransmit to create a folder containing the files to be transmitted and copy the folder to a shared server.
- Package the files using a commercial Unicode-compliant application.
- Make sure all file names use hexidecimal values below 80 before using eTransmit.

To create a transmittal package in a folder you specify

1. Click Application menu  ➤ Publish ➤ eTransmit. 
2. In the Create Transmittal dialog box, Files Tree or Files Table tab, click Add File.
3. (Optional) In the Add File to Transmittal dialog box, locate and select the files that you want to include. Click Open. Repeat this step for additional files, if necessary.
4. In the Create Transmittal dialog box, in the tree or table of files, click to clear the check marks beside any files that you do not want to include. You can click any drawing file node to expand it and display its dependent files.
5. Click Transmittal Setups. Click Modify.
6. In the Modify Transmittal Setup dialog box, click the arrow to display the list under Transmittal Package Type, and then select Folders (Set of Files). Also, specify any additional transmittal options that you want to use.

7 Under Transmittal File Folder, click Browse to specify the folder where the transmittal package should be created. A standard file selection dialog box is displayed.

8 Locate the folder where you want to create the transmittal package. Click Open.

9 Click OK to close the Modify Transmittal Setup dialog box.

10 Click Close to close the Transmittal Setups dialog box.

11 (Optional) In the Create Transmittal dialog box, in the area for notes, specify any additional comments to include with the report file.

12 Click OK to create the transmittal package in the folder you specified.

To create a transmittal package that is a self-extracting executable or ZIP file

1 Click Output tab ➤ Send panel ➤ eTransmit. 

2 In the Create Transmittal dialog box, Files Tree or Files Table tab, click Add File.

3 (Optional) In the Add File to Transmittal dialog box, locate and select the files that you want to include. Click Open. Repeat this step for additional folder locations, if necessary.

4 In the Create Transmittal dialog box, in the tree or table of files, click to clear the check marks beside any files that you do not want to include. On the Files Tree tab, you can click any drawing file node to expand it and display its dependent files.

5 Click Transmittal Setups. Click Modify.

NOTE It is recommended that you create and save several transmittal setups. Then, when you need to create a transmittal package, you can select a transmittal setup rather than following steps 5 through 11.

6 In the Modify Transmittal Setup dialog box, click the arrow to display the list under Transmittal Package Type. Select either Zip (*.*zip*) or Self-Extracting Executable (*.*exe*). Also, specify any additional transmittal options that you want to use.

7 Under Transmittal File Folder, click Browse to specify the folder where the transmittal package should be created. A standard file selection dialog box is displayed.

8 Locate the folder where you want to create the transmittal package. Click Open.

9 Click OK to close the Modify Transmittal Setup dialog box.

10 Click Close to close the Transmittal Setups dialog box.

11 (Optional) In the Create Transmittal dialog box, in the area for notes, specify any additional comments to include with the report file.

12 Click OK to create the transmittal package in the folder you specified.

To create a transmittal package from a sheet set

1 Open the sheet set that you want to transmit.

2 In the Sheet Set Manager, Sheet List tab, right-click one of the following types of nodes from the Sheets-Tree list:

- **Sheet set (top-level node).** To transmit an entire sheet set.
- **Sheet subset (lower-level node).** To transmit a sheet subset or selected portion of a sheet set.
- **Sheet (lowest level node).** To transmit a selected sheet or a selected portion of a sheet set.

3 On the shortcut menu, click ETransmit.

In the Create Transmittal dialog box, on the Sheets tab, sheets are shown indented underneath another sheet when the indented sheets belong to the same drawing file. Selecting or clearing a sheet automatically selects or clears any indented sheets because transmitting a sheet transmits the entire drawing.

NOTE It is recommended that you create and save several transmittal setups. Then, when you need to create a transmittal package, you can select a transmittal setup rather than following steps 4-11.

4 In the Create Transmittal dialog box, click Transmittal Setups. Click Modify.

5 In the Modify Transmittal Setup dialog box, click the arrow to display the list under Transmittal Package Type, and select the type of transmittal

package that you want to create. Also, specify any additional transmittal options that you want to use. Under Transmittal Options, the first option, Use Organized Folder Structure, is usually recommended. Source root folder is the folder where the Drawing Set file (*.dst*) is stored. Use the Keep Files and Folders As Is option for AutoCAD 2007 eTransmit behavior.

6. Under Transmittal File Folder, click Browse to specify the folder where the transmittal package should be created. A standard file selection dialog box is displayed.
7. Locate the folder where you want to create the transmittal package. Click Open.
8. Click OK to close the Modify Transmittal Setup dialog box.
9. Click Close to close the Transmittal Setups dialog box.
10. (Optional) In the Create Transmittal dialog box, in the area for notes, specify any additional comments to include with the report file.
11. Click OK to create the transmittal package in the folder you specified.

To unpack a self-extracting executable transmittal package

1. In Windows Explorer, double-click the transmittal package file.
2. In the eTransmit dialog box, click Browse to specify the folder where you want to unpack the transmittal files.
3. Click OK.
4. If the transmittal package is password protected, you are prompted to enter the password.
5. Click OK to close each dialog box.

Quick Reference

Commands

ARCHIVE

Packages the current sheet set files for archive.

ETRANSMIT

Packages a set of files for Internet transmission.

HYPERLINK

Attaches a hyperlink to an object or modifies an existing hyperlink.

Review and Markup Files with Design Review

Autodesk® Design Review is a free program used for creating and reviewing DWF files. 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.

Design Review enables your entire project or product team to view, print, measure, and markup DWF, DWG, DXF, PDF and raster files containing 2D and 3D content. Fully integrated with AutoCAD®, Inventor®, and Revit®, Design Review helps you easily share drawings, models, maps, and design data with team members, clients, consultants, contractors, partners, suppliers, and other reviewers who may not own or know how to use design software.

You can share designs for use with Design Review by email, websites, intranets, and physical media, such as DVDs. Download Design Review for free (*http://www.autodesk.com/designreview-download*). You can redistribute it on your internal network or deploy it as part of your corporate PC image (as long as it is distributed in its entirety, per the terms of the license agreement).

About DWF and DWFx

A DWF file can be used to organize sheet sets, models, animations, finite element analyses (FEA), and map information, as well as other project-related files, into a single, highly compressed file. Together with Design Review, DWF files help you enhance collaboration by clearly communicating information, such as design changes or corrections, all while reducing the printing and shipping costs associated with distributing paper copies to your extended team.

Much like Adobe® PDF files, DWF files are no more alterable than printed paper copies. Unlike PDF files, however, DWF files retain detailed design information and scale, and are therefore more suitable for architects, engineers, and designers.

The newest version of the DWF file format, DWFx, is based on the XML Paper Specification (XPS) from Microsoft. DWFx makes it easier to share design data with reviewers who cannot install software.

DWFx files can be opened and printed instantly using the free Microsoft XPS Viewer, which comes pre-installed on computers using the Microsoft Windows Vista® operating system. (For the Windows XP operating system, the Microsoft

XPS Viewer can be downloaded directly from Microsoft.) Unlike DWF files, DWFx files include additional information to display design data in the Microsoft XPS Viewer. As such, DWFx files are larger than corresponding DWF files.

TIP In Design Review, you can choose between DWFx and DWF as the default file format on the General tab in the Options dialog box.

Currently, the Microsoft XPS Viewer does not support sheets containing 3D content, password-protected content, object properties, restricted content, or georeferenced map coordinates. In the Microsoft XPS Viewer, when attempting to view sheets DWFx files containing any of these unsupported features, a warning directs you to download and view the DWFx file in Design Review.

NOTE All references to DWF in this documentation implicitly include DWFx, unless specified.

A Digital Design Workflow

Most DWF files begin as a drawing or model created in such Autodesk programs as AutoCAD, Inventor, and Revit. Before a DWF file is published, the person publishing the DWF file determines which features (model, layouts, layers, blocks, named views, and so on) are included in the published DWF file. Once the content has been determined, the designer publishes the file from its original format to a DWF file and sends the DWF file to the review team to begin the digital design review process.

- **Receive.** Reviewers get the DWF file from the publisher and open it in Design Review to verify the content (a 2D drawing, 3D model, or image).
- **Review.** Reviewers add digital comments and markup to the DWF file using callouts, text, shapes, dimensions, stamps, and custom symbols, saving changes to the DWF file.
- **Return.** Reviewers send the marked-up DWF file back to the original publisher.
- **Revise.** The designer uses the publishing software to import the marked-up DWF file, referring to comments in context to revise the original design quickly.
- **Republish.** After revising the original content in the publishing software, the designer republishes an updated DWF file, a new sheet set, or model, to begin the digital design workflow again.

The digital workflow can be repeated indefinitely to support the iterative nature of the design and review process.

Use the Publish to Web Wizard to Create Web Pages

The Publish to Web wizard simplifies the process of creating DWF or DWFx files and formatting them for display in HTML pages.

The Publish to Web wizard provides a simplified interface for creating formatted web pages that include DWF, DWFx, JPEG, or PNG images of drawings.

- DWF or DWFx format does not compress the drawing file.
- JPEG format uses lossy compression; that is, some data is deliberately discarded to greatly reduce the size of the compressed file.
- PNG (Portable Network Graphics) format uses lossless compression; that is, no original data is sacrificed to reduce the size of the file.

Using Publish to Web, you can quickly and easily create an attractively formatted web page, even if you are not familiar with HTML coding. After creating a web page, you can post it to an Internet or intranet location.

Following are some examples of ways in which you can use the Publish to Web wizard to create a web page:

- **Templates**. You can select one of four templates for the layout of your web page or customize your own template.
- **Themes**. You can apply a theme to the template you've chosen. With themes, you can modify the colors and fonts in your web page.
- **i-drop**. You can activate drag-and-drop capability on your web page. Visitors to your page can drag drawing files into a program session. i-drop files are ideally suited for publishing block libraries to the Internet.

For information about using i-drop to create an i-drop handle on a website, see the i-drop documentation on the Autodesk website at *http://www.autodesk.com/idrop*.

NOTE You can also customize the template you use for your web page. For more information about customizing templates, see Customize a Publish to Web Template in the *Customization Guide*.

See also:

- DesignCenter on page 84

To use the Publish to Web wizard

1. Click File menu ➤ Publish to Web. 
2. Follow the instructions to generate a web page.

Quick Reference

Commands

PUBLISHTOWEB

Creates HTML pages that include images of selected drawings.

Use Autodesk Seek to Add and Share Drawings

With Autodesk Seek, you can share, search, and reuse digital design content.

With Autodesk® Seek you can find and share product design information with the online design community to enhance designs and to meet specific customer needs. It allows designers to search for, download, and integrate generic or manufacturer-specific building products or components and associated design information.

When you are working in a design program, you may want to include products that, for example, meet design standards for Leadership in Energy and Environmental Design (LEED) or the Americans with Disabilities Act (ADA). Autodesk Seek can help you locate such information and products, and get them into your design.

- For more information about LEED green building certification, visit the U.S. Green Building Council website: *http://www.usgbc.org*.
- For more information about ADA standards for accessible design, visit U.S. Department of Justice website: *http://www.usdoj.gov/crt/ada/*.

With Autodesk Seek, you can upload and download AutoCAD drawing files and blocks under various categories. You can also access content posted by manufacturers.

Blocks and drawings are uploaded with their attribute values to the Autodesk Seek website.

The host drawing file uploaded to the Autodesk Seek website does not include any external images, external references (xref), or underlay file references.

NOTE Autodesk Seek is currently available in US English only.

To search for blocks and drawings in Autodesk Seek

1. Click Insert tab ➤ Content panel.
2. Click inside the *Seek design content* text box.
3. Enter one or more keywords in the text box, and press Enter or click the Autodesk Seek button. 

To upload a block or drawing to Autodesk Seek

1. Click Output tab ➤ Autodesk Seek panel ➤ Share with Autodesk Seek. 
2. In the Share with Autodesk Seek dialog box, select a block from the list of blocks, or select the current drawing.
3. Click OK.

   The block or drawing is saved as a temporary file, which is packaged with additional information such as a thumbnail image, and sent to the Autodesk Seek website.

   If the drawing is new or contains unsaved changes, you are prompted to save the file.
4. If prompted, enter a new file name and save the file.

   A progress bar is displayed while the file is prepared and uploaded.
5. Your web browser displays the Autodesk Seek website, after the file is uploaded.

   Enter information about your submission on the web page to complete the Share with Autodesk Seek process.

Quick Reference

Commands

ADCENTER

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

CONTENTEXPLORER

Finds and inserts content such as drawing files, blocks, and styles.

CONTENTEXPLORERCLOSE

Closes the Content Explorer window.

SEEK

Opens a web browser and displays the Autodesk Seek home page.

SHAREWITHSEEK

Uploads a selected block or the current drawing to the Autodesk Seek website.

System Variables

CONTENTEXPLORERSTATE

Indicates whether the Content Explorer window is open or closed.

Use Markups for Design Review

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When you are in the final stages of a design, you can publish drawings for review, and receive corrections and comments electronically. You can then implement and respond to these comments and republish your drawings. Doing all this electronically streamlines communication, shortens review cycles, and results in a more efficient design process.

Overview of Using Markups for Design Review

You can use only Autodesk Design Review to mark up DWF or DWFx files for review. You can then view these markups in AutoCAD and make changes to your drawings and change the status of the markups.

When you want to submit designs for review, you can publish a DWF or DWFx file, and then send it to your client or anyone you want to review your designs. The reviewer can open the DWF or DWFx file in Autodesk Design Review, mark up the file electronically, and then send it back to you. For more information on Autodesk Design Review, see Review and Markup Files with Design Review on page 2119.

When you open the marked-up DWF or DWFx file in AutoCAD®, you can view the redline markups in the AutoCAD drawing area, turn them off, make your changes, change the status of the markups, and republish the DWF or DWFx file to send out for another review.

See also:

- Review and Markup Files with Design Review on page 2119

Quick Reference

Commands

MARKUP

Opens the Markup Set Manager.

MARKUPCLOSE

Closes the Markup Set Manager.

OPENDWFMARKUP

Opens a DWF or DWFx file that contains markups.

Publish Drawings for Review

Publish drawings for review at the end of the design cycle.

You should publish drawings for review at the end of the design cycle. In order for electronic markups created in Autodesk Design Review to align with the original drawing geometry, it is important not to change elements in your drawing file that can affect this alignment *after* you have published the DWF or DWFx file for review.

These elements include the following:

- Layers and layer states
- Layers to which objects are tied
- Number and placement of viewports in layouts
- Position and scale of objects within viewports
- View rotation
- Most drawing elements

See also:

- Review and Markup Files with Design Review on page 2119

Quick Reference

Commands

PUBLISH

Insert Markups

Use Autodesk Design Review to insert markups into a DWF or DWFx file. A markup is a single comment or a redline geometry correction inserted into a DWF or DWFx file.

Using Autodesk Design Review, you can add markups electronically, just as you would mark up a paper version of the drawing. You can also assign a status to the markup and add comments.

The markups and their status are saved in the DWF or DWFx file.

For more information about creating markups in Autodesk Design Review, see the Autodesk Design Review Help documentation.

See also:

- Review and Markup Files with Design Review on page 2119

View Markups

Use the Markup Set Manager to view and manage markups. To view the markups in your designs, load the markup set into the Markup Set Manager.

A markup set is a group of markups contained within a single DWF or DWFx file. When you load a markup set into the Markup Set Manager, the tree view displays each marked-up drawing sheet and its associated markups.

In the Markup Set Manager, you can select individual markups, and you can view their status and other details, such as the creator of the markup, the date and time that it was created, and any comments associated with the markup.

You can create other types of markups that will also appear in the DWF or DWFx file: text blocks, redline geometry, dimensions, and stamps. However, only comments have a status and are shown in the Markup Set Manager.

With Autodesk Design Review, you can modify DWF or DWFx files in the following ways:

- Reorder sheets within a DWF or DWFx file

- Add sheets to a DWF or DWFx file

  NOTE Markup Set Manager displays the Sheet status (in Detail view) as "File is missing", if you add sheets using Autodesk Design Review. The reason is that the newly added sheets are not listed in the DSD file.

- Delete sheets from a DWF or DWFx file

Markup Set Manager will reflect the new order for every markup set that has sheets added to, deleted from, or rearranged from within Autodesk Design Review.

Any sheets that were added to the DWF or DWFx file in Autodesk Design Review are listed in italics. When you double-click either a sheet that was added in Autodesk Design Review or a markup for that sheet, that sheet in the DWF or DWFx file will open in the DWF file viewer.

If you try to load a markup set by opening a DWF or DWFx file that does not contain markups, you will not be able to open the DWF or DWFx file in AutoCAD. You can, however, open the DWF or DWFx file in the DWF file viewer. While loading a DWFx file, if the file is digitally signed you will be notified that making changes to the file will invalidate the attached digital signature.

When you double-click a model space markup in the Markup Set Manager, the original drawing file opens in model space and reverts to the view rotation that was current when the drawing was published. If you then change the view rotation, the markups will not be shown in the drawing until you double-click a model space markup and revert to the published view.

Using the Markup Set Manager, you can specify what is displayed in the drawing area. You can show or hide the originating drawing file, the reviewed DWF or DWFx file geometry, and the redline geometry. This allows you to view the redline geometry, turn it off to make the changes in your drawing file, and then view the redline geometry again in AutoCAD to check your corrections.

Following are some recommendations for viewing markups:

- Change the drawing's model space background color to a light shade of a color other than black. This action enables you to view the DWF or DWFx geometry in AutoCAD when the marked-up DWF or DWFx file is returned to you.

- Put the drawing files you used to publish the DWF or DWFx file in the search path. This action ensure that you can load the corresponding drawing sets for a marked-up DWF or DWFx file.
  Set the search path using the Files tab in the Options dialog box.

NOTE You can view markups in AutoCAD only if the marked-up DWF or DWFx file was originally created in AutoCAD 2005 or later. You must also have the original drawing file to view its markups in AutoCAD. Marked-up DWF or DWFx files originally created from earlier versions of AutoCAD can be opened in Autodesk Design Review.

See also:

- Review and Markup Files with Design Review on page 2119

To open the Markup Set Manager

- Click View tab ➤ Palettes panel ➤ Markup Set Manager. 

To open a markup set

1. To open a markup set, do one of the following:
   - Click File menu ➤ Load Markup Set.
   - In the Markup Set Manager, click the Markup Set list control. Click Open.
2. In the Open Markup DWF dialog box, select a DWF or DWFx file that contains markups. Click Open.

   NOTE If you open a digitally signed DWFx file, click Open File in the DWFx - Digital Signature Warning dialog box.

   The Markup Set Manager displays the markup set in the tree view.

   Use Autodesk Design Review to open a DWF or DWFx file that does not contain markups, as AutoCAD will not load non-marked up files.

To view details of an individual markup in the Markup Set Manager

1. Click File menu ➤ Load Markup Set.

2 In the Open Markup DWF dialog box, select a DWF or DWFx file that contains markups. Click Open.

NOTE If you open a digitally signed DWFx file, click Open File in the DWFx - Digital Signature Warning dialog box.

The Markup Set Manager displays the markup set in the tree view.

3 In the Markup Set Manager, click an individual markup node.

The Markup Details area in the lower portion of the Markup Set Manager displays details for the selected markup.

To open a drawing sheet with markups from the Markup Set Manager

1 Click File menu ➤ Load Markup Set.

2 In the Open Markup DWF dialog box, select a DWF or DWFx file that contains markups. Click Open.

NOTE If you open a digitally signed DWFx file, click Open File in the DWFx - Digital Signature Warning dialog box.

The Markup Set Manager displays the markup set in the tree view.

3 In the Markup Set Manager, double-click a drawing sheet or markup node.

If you double-click a markup node, the drawing sheet that corresponds to the markup opens in the drawing area. You must have the original DWG file from which the marked-up DWF or DWFx file was created to view the associated drawing sheet.

NOTE To view markups in the drawing area — visual style and SHADEMODE must be set to 2D Wireframe.

To show or hide DWG geometry for drawing sheets with associated markups

1 Click File menu ➤ Load Markup Set.

2 In the Open Markup DWF dialog box, select a DWF or DWFx file that contains markups. Click Open.

NOTE If you open a digitally signed DWFx file, click Open File in the DWFx - Digital Signature Warning dialog box.

The Markup Set Manager displays the markup set in the tree view.

3 In the Markup Set Manager, double-click a drawing sheet node to open the original DWG file.

4 Click the View DWG Geometry button to hide the DWG geometry. You can click the same button again to show the DWG geometry.

NOTE If you open a DWG file that does not have associated markups or the markup set has not been loaded into the Markup Set Manager, you will not be able to use the following buttons in the Markup Set Manager: Republish All Markup Sets, View DWG Geometry, View DWF or DWFx Geometry, View Redline Geometry.

To show or hide DWF file geometry in the drawing area

1 Click File menu ➤ Load Markup Set.

2 In the Open Markup DWF dialog box, select a DWF or DWFx file that contains markups. Click Open.

NOTE If you open a digitally signed DWFx file, click Open File in the DWFx - Digital Signature Warning dialog box.

The Markup Set Manager opens and displays the markup set in the tree view.

3 In the Markup Set Manager, double-click a drawing sheet node to open the original DWG file.

4 In the Markup Set Manager, click the View DWF Geometry button to show the DWF or DWFx geometry. You can click the same button again to hide the DWF or DWFx geometry.

To show or hide redline geometry (markups) in the drawing area

1 Click File menu ➤ Load Markup Set.

2 In the Open Markup DWF dialog box, select a DWF or DWFx file that contains markups. Click Open.

NOTE If you open a digitally signed DWFx file, click Open File in the DWFx - Digital Signature Warning dialog box.

The Markup Set Manager opens and displays the markup set in the tree view.

3 In the Markup Set Manager, click a markup node to view the redline geometry in the drawing area.

4 Click the View Redline Geometry button to hide the redline geometry. You can click the same button again to show the redline geometry.

To print a drawing with markups

- In Windows Explorer, double-click a DWF or DWFx file that contains markups. The DWF or DWFx file opens in Autodesk Design Review. You can then print the marked-up DWF or DWFx file.

To close the Markup Set Manager

- With the Markup Set Manager open, click Tools menu ➤ Palettes ➤ Markup Set Manager.

Quick Reference

Commands

MARKUP

Opens the Markup Set Manager.

MARKUPCLOSE

Closes the Markup Set Manager.

OPENDWFMARKUP

Opens a DWF or DWFx file that contains markups.

Respond to Markups

Once you have reviewed markups, you can change their status and add comments that are saved with the markup.

In the Markup Set Manager, when you select an individual markup in the tree view, the Markup Details area displays information about the markup. You can change the status of the markup in this area, or you can right-click a markup and click an option on the Markup Status shortcut menu.

The default status for new markups is <None>. You can change a markup's status to Question, For Review, or Done.

In the Markup Set Manager, in the Notes area under Markup Details, you can add comments or notes for the selected markup.

Changes in markup status and added comments are automatically saved in the DWF or DWFx file and included when you republish it. You can also save markup changes by right-clicking the markup set node and clicking Save Markup History Changes on the shortcut menu.

WARNING Saving changes to a digitally signed DWFx file will invalidate the digital signature.

See also:

- Review and Markup Files with Design Review on page 2119

To change the status of a markup

1. Click File menu ➤ Load Markup Set.
2. In the Open Markup DWF dialog box, select a DWF or DWFx file that contains markups. Click Open.

   The Markup Set Manager displays the markup set in the tree view.
3. Right-click a markup node. Click Markup Status and click a status to select it.

   Added comments are automatically saved in the DWF or DWFx file and included when you republish the DWF or DWFx file. You can also save markup changes by right-clicking the markup set node. Click Save Markup History Changes.

To add comments to a markup

1. Click File menu ➤ Load Markup Set.
2. In the Open Markup DWF dialog box, select a DWF or DWFx file that contains markups. Click Open.

   The Markup Set Manager displays the markup set in the tree view.
3. In the Markup Set Manager, click a markup node.
4. In the Details pane, click the Notes text box. Add your comments.

   Comments are saved in the DWF or DWFx file and are included in the file when you republish it.

You can also save markup changes by right-clicking the markup set node. Click Save Markup History Changes.

To save changes to markups

1. Click File menu ➤ Load Markup Set.
2. In the Open Markup DWF dialog box, select a DWF or DWFx file that contains markups. Click Open.

   The Markup Set Manager opens and displays the markup set in the tree view.
3. In the Markup Set Manager, click a markup node and change its status or add comments.
4. Right-click the markup set node. Click Save Markup History Changes.

   Changes to Markup status and any additional comments are also saved in the DWF or DWFx file when you close the drawing or when you republish it.

Quick Reference

Commands

MARKUP

Opens the Markup Set Manager.

MARKUPCLOSE

Closes the Markup Set Manager.

OPENDWFMARKUP

Opens a DWF or DWFx file that contains markups.

Republish a Markup Set

After you have viewed a markup set and made changes to your drawings, you can republish the markup set for another review.

When you republish the markup set, the markups and any changes that have been made to them are saved in the DWF or DWFx file. When anyone reviews the republished DWF or DWFx file in Autodesk Design Review, the changes

made to the drawings are visible as well as any changes made to the status and the details of the markups.

If you republish a DWF or DWFx file that has had sheets deleted from it or rearranged within it in Autodesk Design Review, the new order of sheets will be reflected in the republished DWF or DWFx file; however, if any sheets were added to the DWF or DWFx file in Autodesk Design Review, those sheets will not be included in the republished DWF or DWFx file.

See also:

- Review and Markup Files with Design Review on page 2119

To republish a marked-up drawing set

1. Click File menu ➤ Load Markup Set.
2. In the Open Markup DWF dialog box, select a DWF or DWFx file that contains markups. Click Open.

   The Markup Set Manager opens and displays the markup set in the tree view.
3. In the Markup Set Manager, double-click a markup node or drawing sheet node to open your original DWG file.
4. In the drawing area, make the required changes to your DWG file.
5. In the Markup Set Manager, click markup nodes and change the status or add comments as necessary.
6. Click the Republish Markup DWF button at the top of the Markup Set Manager.
7. Click one of the following:
   - **Republish All Sheets.** Clicking this option republishes all sheets in the marked-up DWF or DWFx file.
   - **Republish Markup Sheets.** Clicking this option republishes only those sheets in the marked-up DWF or DWFx file that have associated markups.
8. In the Specify DWF File dialog box, select a DWF or DWFx file or enter a name for the file. Click Select.

   By default, the previously created marked-up file name is displayed and is overwritten with a file of the same name that contains the drawing and markup changes.

Quick Reference

Commands

MARKUP

Opens the Markup Set Manager.

MARKUPCLOSE

Closes the Markup Set Manager.

OPENDWFMARKUP

Opens a DWF or DWFx file that contains markups.

Part 13: Render Drawings

Draw 2D Isometric Views 43

The Isometric Snap/Grid mode helps you create 2D isometric images that represent 3D objects.

The Isometric Snap/Grid mode helps you create 2D images that represent 3D objects. By setting the Isometric Snap/Grid, you can easily align objects along one of three isometric planes; however, although the isometric drawing appears to be 3D, it is actually a 2D representation. Therefore, you cannot expect to extract 3D distances and areas, display objects from different viewpoints, or remove hidden lines automatically.

Set Isometric Grid and Snap

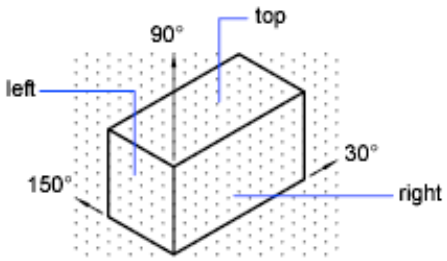
Simulate a 3D object from a particular viewpoint by aligning along three major axes.

Isometric drawings simulate a 3D object from a particular viewpoint by aligning along three major axes.

By setting the Isometric Snap/Grid, you can easily align objects along one of three isometric planes; however, although the isometric drawing appears to be 3D, it is actually a 2D representation. Therefore, you cannot expect to extract 3D distances and areas, display objects from different viewpoints, or remove hidden lines automatically.

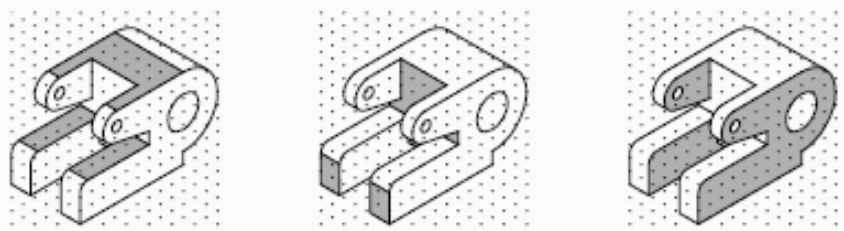
If the snap angle is 0, the axes of the isometric planes are 30 degrees, 90 degrees, and 150 degrees. Once you set the snap style to Isometric, you can work on any of three planes, each with an associated pair of axes:

- **Left.** Aligns snap and grid along 90- and 150-degree axes.
- **Top.** Aligns snap and grid along 30- and 150-degree axes.
- **Right.** Aligns snap and grid along 30- and 90-degree axes.



isometric drawing planes

Choosing one of the three isometric planes causes Ortho and the crosshairs to be aligned along the corresponding isometric axes. For example, when Ortho is on, the points you specify align along the simulated plane you are drawing on. Therefore, you can draw the top plane, switch to the left plane to draw another side, and switch to the right plane to complete the drawing.



To turn on an isometric plane

1. Click Tools menu ➤ Drafting Settings.
2. In the Drafting Settings dialog box, Snap and Grid tab, under Snap Type, select Isometric Snap.
3. Click OK.

Alternately, you can cycle through the three isometric planes by pressing F5.

To change to a different isometric plane quickly

- Press F5 or CTRL+E to select a different isometric plane.

The program cycles through the Isoplane Top, Isoplane Right, and Isoplane Left 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.

ISOPLANE

Specifies the current isometric plane.

ORTHO

Constrains cursor movement to the horizontal or vertical direction.

SNAP

Restricts cursor movement to specified intervals.

System Variables

SNAPISOPAIR

Controls the isometric plane for the current viewport.

SNAPSTYL

Sets the snap style for the current viewport.

Draw Isometric Circles

Represent circles on isometric planes using ellipses.

If you are drawing on isometric planes, use an ellipse to represent a circle viewed from an oblique angle. The easiest way to draw an ellipse with the correct shape is to use the Isocircle option of ELLIPSE. The Isocircle option is available only when the Style option of Snap mode is set to Isometric (see DSETTINGS).

NOTE To represent concentric circles, draw another ellipse with the same center rather than offsetting the original ellipse. Offsetting produces an oval-shaped spline that does not represent foreshortened distances as you would expect.

To draw an isometric circle

1. Click Tools menu ➤ Drafting Settings.
2. In the Drafting Settings dialog box, Snap and Grid tab, select Isometric Snap.
3. Click OK.
4. Click Draw ➤ Ellipse ➤ Axis, End.
5. Enter **i** (Isocircle).
6. Specify the center of the circle.
7. Specify the radius or diameter of the circle.

Quick Reference

Commands

ELLIPSE

Creates an ellipse or an elliptical arc.

ISOPLANE

Specifies the current isometric plane.

Add Lighting to Your Model

44

Lighting can be added to a scene to create a more realistic rendering.

Overview of Lighting

Lighting adds the finishing touch to the scene.

Default Lighting

When there are no lights in a scene, the scene is shaded with default lighting. Default lighting is derived from two distant sources that follow the viewpoint as you move around the model. All faces in the model are illuminated so that they are visually discernible. You can control brightness and contrast, but you do not need to create or place lights yourself.

When you insert custom lights or add sunlight, you can disable the default lighting. You can apply default lighting to the viewport only; at the same time, you can apply custom lights to the rendering.

Standard Lighting Workflow

You add lights to give the scene a realistic appearance. Lighting enhances the clarity and three-dimensionality of a scene. You can create point lights, spotlights, and distant lights to achieve the effects you want. You can move or rotate them with grip tools, turn them on and off, and change properties such as color and attenuation. The effects of changes are visible in the viewport in real time.

Spotlights and point lights are each represented by a different light glyph (a symbol in the drawing showing the location of the light). Distant lights and the sun are not represented by glyphs in the drawing because they do not have

a discrete position and affect the entire scene. You can turn the display of light glyphs on or off while you work. By default, light glyphs are not plotted.

Photometric Lighting Workflow

For more precise control over lighting, you can use photometric lights to illuminate your model. Photometric lights use photometric (light energy) values that enable you to define lights more accurately as they would be in the real world. You can create lights with various distribution and color characteristics, or import specific photometric files available from lighting manufacturers.

Photometric lights can use manufacturers' IES standard file format. By using manufacturers' lighting data, you can visualize commercially available lighting in your model. Then you can experiment with different fixtures, and by varying the light intensity and color temperature, you can design a lighting system that produces the results you want.



Sun and Sky

The sun is a special light similar to a distant light. The angle of the sun is defined by the geographic location that you specify for the model and by the date and time of day that you specify. You can change the intensity of the sun and the color of its light. The sun and sky are the primary sources of natural illumination. With the sun and sky simulation on page 2194, you can adjust their properties.

In the photometric workflow, the sun follows a more physically accurate lighting model in both the viewport and the rendered output. In the photometric workflow, you can also enable sky illumination (through the sky

background feature), which adds soft, subtle lighting effects caused by the lighting interactions between the sun and the atmosphere.

Luminaire Objects

Light fixtures can be represented by embedding photometric lights in blocks that also contain geometry. A luminary assembles a set of light objects into a light fixture.



Quick Reference

Commands

CONVERTOLDLIGHTS

Converts lights created in previous drawing file formats to the current format.

DISTANTLIGHT

Creates a distant light.

GEOGRAPHICLOCATION

Specifies the geographic location information for a drawing file.

LIGHT

Creates a light.

LIGHTLIST

Turns on and off the Lights in Model palette that lists all lights in the model.

FREESPOT

Creates free spotlight which is similar to a spotlight without a specified target.

FREEWEB

Creates a free web light which is similar to a web light without a specified target.

POINTLIGHT

Creates a point light that radiates light in all directions from its location.

RENDEREXPOSURE

Provides settings to adjust the global lighting for the most recently rendered output.

SPOTLIGHT

Creates a spotlight that emits a directional cone of light.

SUNPROPERTIES

Displays the Sun Properties window.

TARGETPOINT

Creates a target point light.

WEBLIGHT

Creates a web light.

System Variables

3DCONVERSIONMODE

Used to convert material and light definitions to the current product release.

DEFAULTLIGHTING

Turns on and off default lighting in place of other lighting.

DEFAULTLIGHTINGTYPE

Specifies the type of default lighting, old or new.

LIGHTGLYPHDISPLAY

Turns on and off the display of light glyphs.

LIGHTINGUNITS

Controls whether generic or photometric lights are used, and specifies the lighting units for the drawing.

LIGHTSINBLOCKS

Controls whether lights contained in blocks are used when rendering.

LINEARBRIGHTNESS

Controls the brightness level of the viewport when using default lighting or generic lights.

LINEARCONTRAST

Controls the contrast level of the viewport when using default lighting or generic lights.

LOGEXPBRIGHTNESS

Controls the brightness level of the viewport when using photometric lighting.

LOGEXPCONTRAST

Controls the contrast level of the viewport when using photometric lighting.

LOGEXPDAYLIGHT

Controls if the exterior daylight flag is enabled when using photometric lighting.

LOGEXPMIDTONES

Controls the mid tones level of the viewport when using photometric lighting.

RENDERUSERLIGHTS

Controls whether to override the setting for viewport lighting during rendering.

SUNSTATUS

Turns on and off the lighting effects of the sun in the current viewport.

Standard and Photometric Lighting Workflow

Types of lighting selected globally affect a drawing.

Set the Type of Lighting

AutoCAD offers three choices for lighting units: standard (generic), International (SI), and American. The standard (generic) lighting workflow is equivalent to the lighting workflow in AutoCAD prior to AutoCAD 2008. The default lighting workflow for drawings created in AutoCAD 2008 and later is a photometric workflow based on International (SI) lighting units. This choice results in physically correct lighting. The American lighting unit provides another option. American differs from International in that illuminance values are formatted in foot-candles rather than lux.

In previous versions of AutoCAD, standard lighting was the default. You can change the type of lighting with the LIGHTINGUNITS system variable. The LIGHTINGUNITS system variable set to 0 represents standard (generic) lighting; set to 1 represents photometric lighting in American units; set to 2 represents photometric lighting in International SI units.

Photometric Lights

Photometric lights are physically correct lights. Physically correct lights attenuate as the square of the distance.

Photometric properties can be added to both artificial lights and natural lights. Natural lights are the sun and the sky. The natural lighting is represented interactively by a viewport background type.

You can create lights with various distribution and color characteristics, or import specific photometric files available from lighting manufacturers. Photometric lights always attenuate using an inverse-square falloff, and rely on your scene to use realistic units.

Tone Mapping

Generally with photometric lights and especially the sun you need to perform tone mapping. Tone mapping can be adjusted with the RENDEREXPOSURE command. The Adjust Rendered Exposure dialog box provides a preview and controls to adjust the tone mapping.

To set the drawing for standard lighting workflow

Do one of the following:

- Click Render tab ➤ Lights panel ➤ Lighting Units drop-down ➤ Generic Lighing Units. 
  No lighting units are used and standard (generic) lighting is enabled.

- At the command prompt, enter LIGHTINGUNITS and set the value to 0 for standard lighting.
  As each light is added it will have standard lighting properties.

To set the drawing for photometric workflow

Do one of the following:

- Click Render tab ➤ Lights panel ➤ Lighting Units drop-down ➤ International Lighing Units. 
  International lighting units are used and photometric lighting is enabled.
- Click Render tab ➤ Lights panel ➤ Lighting Units drop-down ➤ American Lighing Units. 
  American lighting units are used and photometric lighting is enabled.
- At the command prompt, enter LIGHTINGUNITS and set the value to 1 (American units) or 2 (International SI units) for photometric lighting.
  As each light is added it will have photometric lighting properties.

Quick Reference

System Variables

LINEARBRIGHTNESS

Controls the brightness level of the viewport when using default lighting or generic lights.

LINEARCONTRAST

Controls the contrast level of the viewport when using default lighting or generic lights.

LOGEXPBRIGHTNESS

Controls the brightness level of the viewport when using photometric lighting.

LOGEXPCONTRAST

Controls the contrast level of the viewport when using photometric lighting.

LOGEXPDAYLIGHT

Controls if the exterior daylight flag is enabled when using photometric lighting.

LOGEXPMIDTONES

Controls the mid tones level of the viewport when using photometric lighting.

LOGEXPPHYSICALSCALE

Controls the relative brightness of self-illuminated materials in a photometric environment.

Illuminate a Scene

You can add point lights, spotlights, and distant lights and set the location and photometric properties of each.

You can use a command to create a light, or you can use a button on the Lights toolbar or the Lights panel on the ribbon. You can use the Properties palette to change the color of a selected light or other properties. You can also store a light and its properties on a tool palette and use it again in the same drawing or another drawing.

Guidelines for Lighting

The guidelines for lighting used by photographers, filmmakers, and stage designers can help you set up the lighting for scenes.

Your choice of lighting depends on whether your scene simulates natural or artificial illumination. Naturally lit scenes, such as daylight or moonlight, get their most important illumination from a single light source. Artificially lit scenes, on the other hand, often have multiple light sources of similar intensity.

Natural Light

For practical purposes at ground level, sunlight has parallel rays coming from a single direction. The direction and angle vary depending on the time of day, the latitude, and the season.

In clear weather, the color of sunlight is a pale yellow: for example, RGB values of 250, 255, 175 (HSV 45, 80, 255). Cloudy weather can tint sunlight blue, shading into dark gray for stormy weather. Particles in the air can give sunlight

an orange or brownish tint. At sunrise and sunset, the color can be more orange or red than yellow.

Shadows are more distinct the clearer the day is, and can be essential for bringing out the three-dimensionality of a naturally lit scene.

A directional light can also simulate moonlight, which is white but dim compared to the sun.

Artificial Light

A scene illuminated by point lights, spotlights, or distant lights is artificially illuminated. Therefore, it can be helpful to know how light behaves.

When light rays strike a surface, the surface reflects them, or at least some of them, enabling us to see the surface. The appearance of a surface depends on the light that strikes it combined with the properties of the surface material, such as color, smoothness, and opacity.

Other factors, such as a light's color, intensity, attenuation, and angle of incidence also play a role in how objects in a scene appear.

Quick Reference

Commands

DISTANTLIGHT

Creates a distant light.

FREESPOT

Creates free spotlight which is similar to a spotlight without a specified target.

LIGHT

Creates a light.

LIGHTLIST

Turns on and off the Lights in Model palette that lists all lights in the model.

POINTLIGHT

Creates a point light that radiates light in all directions from its location.

RENDEREXPOSURE

Provides settings to adjust the global lighting for the most recently rendered output.

SPOTLIGHT

Creates a spotlight that emits a directional cone of light.

TARGETPOINT

Creates a target point light.

System Variables

DEFAULTLIGHTING

Turns on and off default lighting in place of other lighting.

DEFAULTLIGHTINGTYPE

Specifies the type of default lighting, old or new.

LIGHTGLYPHDISPLAY

Turns on and off the display of light glyphs.

LIGHTINGUNITS

Controls whether generic or photometric lights are used, and specifies the lighting units for the drawing.

LINEARBRIGHTNESS

Controls the brightness level of the viewport when using default lighting or generic lights.

LINEARCONTRAST

Controls the contrast level of the viewport when using default lighting or generic lights.

RENDERUSERLIGHTS

Controls whether to override the setting for viewport lighting during rendering.

Use Point Lights

A point light radiates light in all directions from its location.

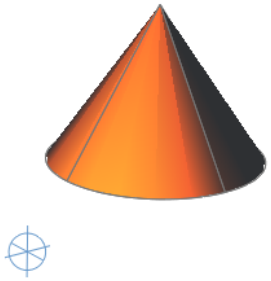
Point Lights

A point light radiates light in all directions from its location. A point light does not target an object. Use point lights for general lighting effects. You can

create a point light by entering the POINTLIGHT command or by selecting a point light from the Lights panel on the ribbon.

You create a target point light with the TARGETPOINT command. The difference between the target point light and a point light is the additional target properties that are available. A target light can be pointed to an object. A target point light can also be created from a point light by changing the target property of the point light from No to Yes.

In the standard lighting workflow, you can set a point light manually so its intensity diminishes with respect to distance either linearly, according to the inverse square of the distance, or not at all. By default, the attenuation is set to None.



Point Lights in Photometric Workflow

A free point light can have photometric distribution properties. The attenuation for a photometric point light is always set to inverse square.

When the LIGHTINGUNITS system variable is set to 1 (American units) or 2 (International SI units) for photometric lighting, additional properties are available for a point light. On the Properties palette, photometric properties are

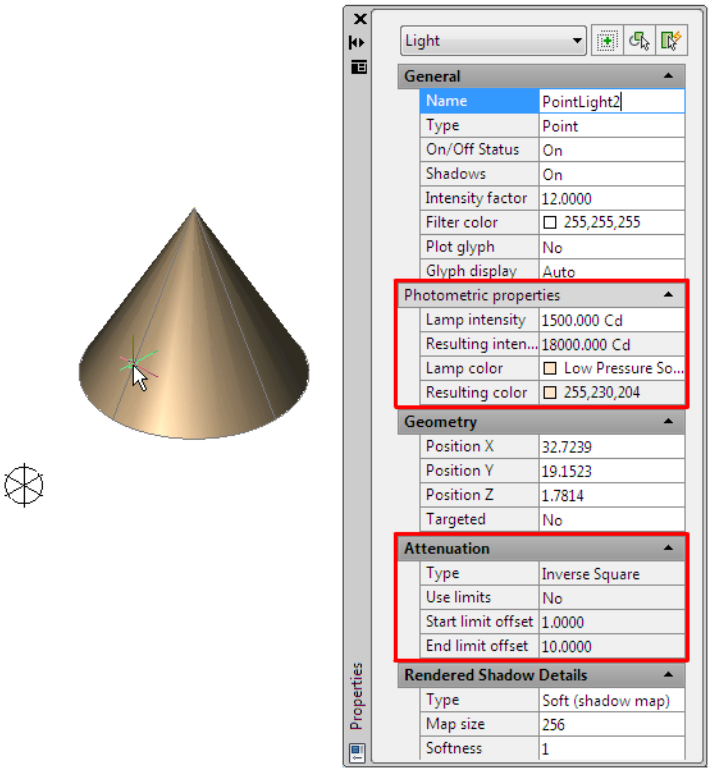
- **Lamp Intensity.** Specifies the inherent brightness of the light. Specifies the intensity, flux or illuminance of the lamp.
- **Resulting Intensity.** Gives the final brightness of the light. (Product of lamp intensity and intensity factor. Read-only.)

- **Lamp Color.** Specifies the inherent color of the light in Kelvin temperature or standard.
- **Resulting Color.** Gives the final color of the light. This is determined by a combination of the lamp color and the filter color. (Product of lamp color and filter color. Read-only.)

NOTE When the drawing lighting units are photometric, the attenuation type property becomes disabled. Photometric lights have fixed, inverse-square attenuation.

The additional information about these properties are available under Lighting Properties.

The following image is an example of a photometric point light and lighting properties palette with the photometric properties outlined:



To create a point light in standard lighting workflow

1. Click Render tab ➤ Lights panel ➤ Lighting Units drop-down ➤ Generic Lighing Units. 

   No lighting units are used and standard (generic) lighting is enabled.

   NOTE Alternatively, you can enter **lightingunits** at the Command prompt and set the value to 0 for standard (generic) lighting.

2. Click Render tab ➤ Lights panel ➤ Create Light drop-down ➤ Point. 

3. Click in the drawing to specify a location for the light.

4. At the Command prompt, enter **n** and enter a name. This name will appear in the properties and in the Lights in Model window (LIGHTLIST).

   You can continue to specify properties by entering options, or you can exit and set properties interactively. When you use the interactive method, you can see the results of your changes as you work.

5. Press Enter twice to exit the command.

   Select the light and use grip tools to change the light. You can also right-click the light and then click Properties. The Lighting Properties window can be used to change its properties.

To create a point light in photometric workflow and change photometric properties

1. Click Render tab ➤ Lights panel ➤ Lighting Units drop-down ➤ International Lighing Units. 

   International lighting units are used and photometric lighting is enabled.

   NOTE Alternatively, you can enter **lightingunits** at the Command prompt and set the value to 2 for international SI lighting.

2 Click Render tab ➤ Lights panel ➤ Create Light drop-down ➤ Point.



3 Click in the drawing to specify a location for the light.

4 At the Command prompt, enter **n** and enter a name. This name will appear in the properties and in the Lights in Model window (LIGHTLIST).

You can continue to specify properties by entering options, or you can exit and set properties interactively. When you use the interactive method, you can see the results of your changes as you work.

5 Press Enter twice to exit the command.

Select the light and use grip tools to change the light.

6 Right-click the light. Click Properties. You can use the Lighting Properties window to change the photometric properties.

To create a target point light in standard lighting workflow

1 Click Render tab ➤ Lights panel ➤ Lighting Units drop-down ➤ Generic Lighing Units. 

No lighting units are used and standard (generic) lighting is enabled.

NOTE Alternatively, you can enter **lightingunits** at the Command prompt and set the value to 0 for standard (generic) lighting.

2 At the command prompt, enter targetpoint.

3 Click in the drawing to specify a location for the light.

4 Click to specify a target point.

5 At the Command prompt, enter **n** and enter a name. This name will appear in the properties and in the Lights in Model window (LIGHTLIST).

You can continue to specify properties by entering options, or you can exit and set properties interactively. When you use the interactive method, you can see the results of your changes as you work.

6 Press Enter twice to exit the command.

Right-click the light. Click Properties. You can use the Lighting Properties palette to change the light. Note that compared to the point light there are additional target positioning properties available.

To create a target point light from a point light

1. Click Render tab ➤ Lights panel ➤ Lighting Units drop-down ➤ Generic Lighing Units. 

   No lighting units are used and standard (generic) lighting is enabled.

   NOTE Alternatively, you can enter **lightingunits** at the Command prompt and set the value to 0 for standard (generic) lighting.

2. Click Render tab ➤ Lights panel ➤ Create Light drop-down ➤ Point. 

3. Click in the drawing to specify a location for the light.

4. At the Command prompt, enter **n** and enter a name. This name will appear in the properties and in the Lights in Model window (LIGHTLIST).

   You can continue to specify properties by entering options, or you can exit and set properties interactively. When you use the interactive method, you can see the results of your changes as you work.

5. Press Enter twice to exit the command.

6. Right-click the light and click Properties on the Lighting Properties palette. Change the Targeted property to **Yes**.

   Additional target properties become available: Target X, Target Y, and Target Z.

   NOTE When changing a point light to target light, the name of the light remains the same. If it still says point light, you might want to change the name to target light.

To create a point light tool

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

2 In the drawing, select a point light whose properties you want to use for the tool.

3 Drag the light to the active tool palette.

The new light tool retains all of the properties of the light except its location properties.

Quick Reference

Commands

LIGHT

Creates a light.

LIGHTLIST

Turns on and off the Lights in Model palette that lists all lights in the model.

POINTLIGHT

Creates a point light that radiates light in all directions from its location.

RENDEREXPOSURE

Provides settings to adjust the global lighting for the most recently rendered output.

TARGETPOINT

Creates a target point light.

System Variables

DEFAULTLIGHTING

Turns on and off default lighting in place of other lighting.

DEFAULTLIGHTINGTYPE

Specifies the type of default lighting, old or new.

LIGHTGLYPHDISPLAY

Turns on and off the display of light glyphs.

LIGHTINGUNITS

Controls whether generic or photometric lights are used, and specifies the lighting units for the drawing.

LIGHTSINBLOCKS

Controls whether lights contained in blocks are used when rendering.

LINEARBRIGHTNESS

Controls the brightness level of the viewport when using default lighting or generic lights.

LINEARCONTRAST

Controls the contrast level of the viewport when using default lighting or generic lights.

LOGEXPBRIGHTNESS

Controls the brightness level of the viewport when using photometric lighting.

LOGEXPCONTRAST

Controls the contrast level of the viewport when using photometric lighting.

LOGEXPDAYLIGHT

Controls if the exterior daylight flag is enabled when using photometric lighting.

LOGEXPMIDTONES

Controls the mid tones level of the viewport when using photometric lighting.

RENDERUSERLIGHTS

Controls whether to override the setting for viewport lighting during rendering.

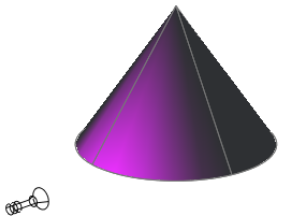
Use Spotlights

A spotlight can be directed towards an object.

Spotlights

A spotlight distribution casts a focused beam of light like a flashlight, a follow spot in a theater, or a headlight. A spotlight emits a directional cone of light. You can control the direction of the light and the size of the cone. Like a point light, a spot light can be manually set to attenuate its intensity with distance. However, a spotlight's intensity will also always attenuate based on the angle relative to the spot's target vector. This attenuation is controlled by the hotspot and falloff angles of the spotlight. Spotlights are useful for highlighting specific

features and areas in your model. A free spotlight (FREESPOT) is similar to spotlight. A spotlight has target properties.



Spotlights in Photometric Workflow

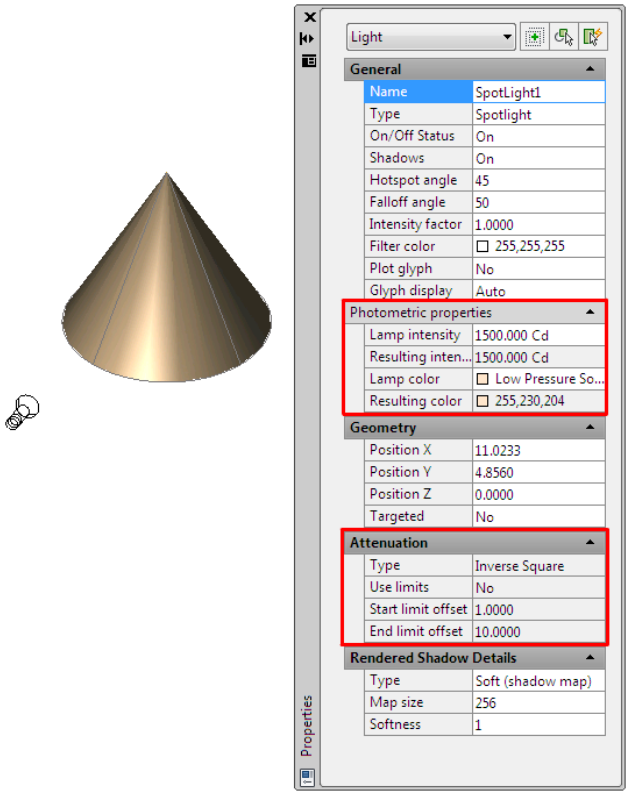
In photometric workflow, the hotspot intensity falls to 50 percent. The hotspot for standard lighting is at 100 percent. At its falloff angle, intensity of the spotlight falls to zero. Additional properties become available for a point light when LIGHTINGUNITS is set to 1 (American units) or 2 (International SI units) for photometric lighting:

- **Lamp Intensity.** Specifies the inherent brightness of the light. Specifies the intensity, flux, or illuminance of the lamp.
- **Resulting Intensity.** Gives the final brightness of the light. (Product of lamp intensity and intensity factor. Read-only.)
- **Lamp Color.** Specifies the inherent color of the light in Kelvin temperature or standard.
- **Resulting Color.** Gives the final color of the light. This is determined by a combination of the lamp color and the filter color. (Product of lamp color and filter color. Read-only.)

NOTE When the drawing lighting units are photometric, the attenuation type property becomes disabled. Photometric lights have fixed, inverse-square attenuation. The hotspot falloff attenuation in the rendered image varies from standard lighting, as it uses a different mathematical basis.

The additional information about these properties are available under Lighting Properties.

The following image is an example of a photometric spotlight and lighting properties palette with the photometric properties outlined:



To create a spotlight

1 Click Render tab ➤ Lights panel ➤ Lighting Units drop-down ➤ Generic Lighing Units. 

No lighting units are used and standard (generic) lighting is enabled.

NOTE Alternatively, you can enter **lightingunits** at the Command prompt and set the value to 0 for standard (generic) lighting.

2 Click Render tab ➤ Lights panel ➤ Create Light drop-down ➤ Spot.



3 Click in the drawing to specify a location for the light.

4 Click to specify a target for the spotlight.

5 At the Command prompt, enter **n** and enter a name. This name will appear in the properties and in the Lights in Model window (LIGHTLIST).

You can continue to specify properties by entering options, or you can exit and set properties interactively. When you use the interactive method, you can see the results of your changes as you work.

6 Press Enter twice to exit the command.

Select the light and use grip tools to change the light. You can also right-click the light and then click Properties. Use Lighting Properties palette to change its properties.

To create a photometric spotlight and change photometric properties

1 Click Render tab ➤ Lights panel ➤ Lighting Units drop-down ➤ Generic Lighing Units. 

No lighting units are used and standard (generic) lighting is enabled.

NOTE Alternatively, you can enter **lightingunits** at the Command prompt and set the value to 0 for standard (generic) lighting.

2 Click Render tab ➤ Lights panel ➤ Create Light drop-down ➤ Spot.



3 Click in the drawing to specify a location for the light.

4 Click to specify a target for the spotlight.

5 At the Command prompt, enter **n** and enter a name. This name will appear in the properties and in the Lights in Model window (LIGHTLIST).

You can continue to specify properties by entering options, or you can exit and set properties interactively. When you use the interactive method, you can see the results of your changes as you work.

6 Press Enter twice to exit the command.

Select the light and use grip tools to change the light.

7 To change properties, right-click the light. Click Properties. Use the Lighting Properties palette to change the photometric properties.

To create a free spot light in photometric lighting workflow

1 At the Command prompt, enter **lightingunits** and set the value to 1 (American units) or 2 (International SI units) for photometric lighting.

2 At the command prompt, enter freespot.

3 Click in the drawing to specify a location for the light.

4 At the Command prompt, enter **n** and enter a name. This name will appear in the properties and in the Lights in Model window (LIGHTLIST).

You can continue to specify properties by entering options, or you can exit and set properties interactively. When you use the interactive method, you can see the results of your changes as you work.

5 Press Enter twice to exit the command.

Select the light and use grip tools to change the light.

6 To change properties, right-click the light. Click Properties. Use the Lighting Properties window to change the photometric properties.

To create a spotlight tool

1 Click Veiw tab ➤ Palettes panel ➤ Tool Palettes. 

2 In the drawing, select a spotlight whose properties you want to use for the tool.

3 Drag the light to the active tool palette.

The new light tool retains all of the properties of the light except its location properties.

Quick Reference

Commands

FREESPOT

Creates free spotlight which is similar to a spotlight without a specified target.

LIGHT

Creates a light.

LIGHTLIST

Turns on and off the Lights in Model palette that lists all lights in the model.

RENDEREXPOSURE

Provides settings to adjust the global lighting for the most recently rendered output.

SPOTLIGHT

Creates a spotlight that emits a directional cone of light.

System Variables

DEFAULTLIGHTING

Turns on and off default lighting in place of other lighting.

DEFAULTLIGHTINGTYPE

Specifies the type of default lighting, old or new.

LIGHTGLYPHDISPLAY

Turns on and off the display of light glyphs.

LIGHTINGUNITS

Controls whether generic or photometric lights are used, and specifies the lighting units for the drawing.

LIGHTSINBLOCKS

Controls whether lights contained in blocks are used when rendering.

LINEARBRIGHTNESS

Controls the brightness level of the viewport when using default lighting or generic lights.

LINEARCONTRAST

Controls the contrast level of the viewport when using default lighting or generic lights.

LOGEXPBRIGHTNESS

Controls the brightness level of the viewport when using photometric lighting.

LOGEXPCONTRAST

Controls the contrast level of the viewport when using photometric lighting.

LOGEXPDAYLIGHT

Controls if the exterior daylight flag is enabled when using photometric lighting.

LOGEXPMIDTONES

Controls the mid tones level of the viewport when using photometric lighting.

RENDERUSERLIGHTS

Controls whether to override the setting for viewport lighting during rendering.

Use Weblights

Weblights are photometric lights with customized, real-world light distributions.

Overview of Weblights

Weblights are photometric lights with customized, real-world light distributions.

A weblight (web) is a 3D representation of the light intensity distribution of a light source. Weblights can be used to represent anisotropic (non-uniform) light distributions derived from data provided by manufacturers of real-world lights. This gives a far more precise representation of the rendered light than either spot or point lights are capable of.

This directional light distribution information is stored in a photometric data file in the IES format using the IES LM-63-1991 standard file format for photometric data. You can load photometric data files provided by various

manufacturers under the Photometric Web panel in the Properties palette for the light. The light icon represents the photometric web you select.

A light that uses a photometric web can be added to a drawing by entering the commands WEBLIGHT and FREEWEB at the command prompt. The WEBLIGHT command creates a targeted weblight, whereas the FREEWEB command creates a weblight without an explicit target.

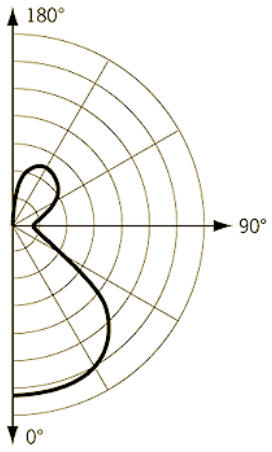
To describe the directional distribution of the light emitted by a source, AutoCAD approximates the source by a point light placed at its photometric center. With this approximation, the distribution is characterized as a function of the outgoing direction only. The luminous intensity of the source for a predetermined set of horizontal and vertical angles is provided, and the system can compute the luminous intensity along an arbitrary direction by interpolation.

NOTE Web distribution is used only in rendered images. Weblights are approximated as point lights in the viewport.

NOTE You can control the size of a web glyph from Tools menu > Options > Drafting > Light Glyph Settings. Some web glyphs may appear very small on screen and may need to be adjusted.

Goniometric Diagrams

Photometric data is often depicted using a goniometric diagram.



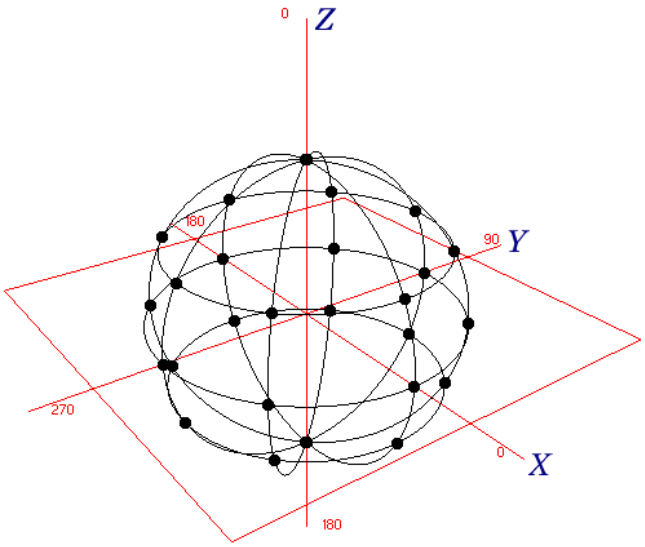
Goniometric diagram of a web distribution

This type of diagram visually represents how the luminous intensity of a source varies with the vertical angle. However, the horizontal angle is fixed and, unless the distribution is axially symmetric, more than one goniometric diagram may be needed to describe the complete distribution.

Photometric Webs

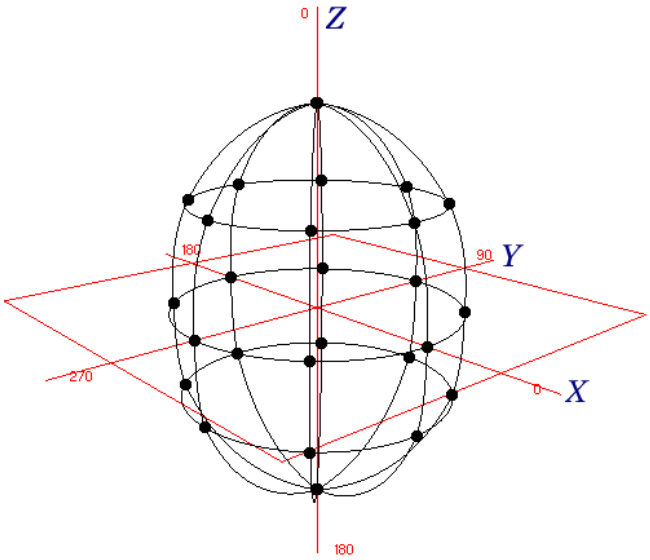
The photometric web is a three dimensional representation of the light distribution. It extends the goniometric diagram to three dimensions, so that the dependencies of the luminous intensity on both the vertical and horizontal angles can be examined simultaneously. The center of the photometric web represents the center of the light object.

The luminous intensity in any given direction is proportional to the distance between this web and the photometric center, measured along a line leaving the center in the specified direction.



Example of Isotropic distribution

A sphere centered around the origin is a representation of an isotropic distribution. All the points in the diagram are equidistant from the center and therefore light is emitted equally in all directions.



Example of Ellipsoidal distribution

In this example, the points in the negative Z direction are the same distance from the origin as the corresponding points in the positive Z direction, so the same amount of light shines upward and downward. No point has a very large X or Y component, either positive or negative, so less light is cast laterally from the light source.

To create a web distribution with a manufacturer's IES file

1. At the command prompt, enter **lightingunits** and set the value to 1 (American units) or 2 (International SI units) for photometric lighting.
2. At the Command prompt, enter properties to activate the Properties palette.
3. Enter **spotlight** at the Command prompt.
4. In the drawing, select the light. The properties for the spotlight are displayed in the Properties palette.
5. Under the General panel, change the lighting Type from spotlight to web.
6. Under the Photometric Web panel, select the blank text box.
7. Click the browse button and then select an IES file.

The light has been changed to have a web distribution with the web settings from the IES file.

Quick Reference

Commands

LIGHT

Creates a light.

LIGHTLIST

Turns on and off the Lights in Model palette that lists all lights in the model.

FREEWEB

Creates a free web light which is similar to a web light without a specified target.

RENDEREXPOSURE

Provides settings to adjust the global lighting for the most recently rendered output.

WEBLIGHT

Creates a web light.

System Variables

DEFAULTLIGHTING

Turns on and off default lighting in place of other lighting.

DEFAULTLIGHTINGTYPE

Specifies the type of default lighting, old or new.

LIGHTGLYPHDISPLAY

Turns on and off the display of light glyphs.

LIGHTINGUNITS

Controls whether generic or photometric lights are used, and specifies the lighting units for the drawing.

LIGHTSINBLOCKS

Controls whether lights contained in blocks are used when rendering.

LINEARBRIGHTNESS

Controls the brightness level of the viewport when using default lighting or generic lights.

LINEARCONTRAST

Controls the contrast level of the viewport when using default lighting or generic lights.

LOGEXPBRIGHTNESS

Controls the brightness level of the viewport when using photometric lighting.

LOGEXPCONTRAST

Controls the contrast level of the viewport when using photometric lighting.

LOGEXPDAYLIGHT

Controls if the exterior daylight flag is enabled when using photometric lighting.

LOGEXPMIDTONES

Controls the mid tones level of the viewport when using photometric lighting.

RENDERUSERLIGHTS

Controls whether to override the setting for viewport lighting during rendering.

SUNSTATUS

Turns on and off the lighting effects of the sun in the current viewport.

IES Standard File Format

IES standard file formats can be created and modified.

You can create a photometric data file in the IES format using the IES LM-63-1991 standard file format for photometric data. (IES stands for Illuminating Engineering Society.) However, only the information relevant to AutoCAD is described here. For a complete description of the IES standard file format, see IES Standard File Format for Electronic Transfer of Photometric Data and Related Information, prepared by the IES Computer Committee (*http://www.iesna.org*).

The luminous intensity distribution (LID) of a luminaire is measured at the nodes of a photometric web for a fixed set of horizontal and vertical angles.

The poles of the web lie along the vertical axis, with the nadir corresponding to a vertical angle of zero degrees. The horizontal axis corresponds to a horizontal angle of zero degrees and is oriented parallel to the length of the luminaire. This type of photometric web is generated by a Type C goniometer and is the most popular in North America; other types of goniometry are supported by the IES standard file format but are not discussed here.

The photometric data is stored in an ASCII file. Each line in the file must be less than 132 characters long and must be terminated by a carriage return/line-feed character sequence. Longer lines can be continued by inserting a carriage return/line-feed character sequence.

Each field in the file must begin on a new line and must appear exactly in the following sequence:

1. IESNA91
2. [TEST] The test report number of your data
3. [MANUFAC] The manufacturer of the luminaire
4. TILT=NONE
5. 1
6. The initial rated lumens for the lamp used in the test or -1 if absolute photometry is used and the intensity values do not depend on different lamp ratings.
7. A multiplying factor for all the candela values in the file. This makes it possible to easily scale all the candela values in the file when the measuring device operates in unusual units—for example, when you obtain the photometric values from a catalog using a ruler on a goniometric diagram. Normally the multiplying factor is 1.
8. The number of vertical angles in the photometric web.
9. The number of horizontal angles in the photometric web.
10. 1
11. The type of unit used to measure the dimensions of the luminous opening. Use 1 for feet or 2 for meters.
12. The width, length, and height of the luminous opening. It is normally given as 0 0 0.
13. 1.0 1.0 0.0

14 The set of vertical angles, listed in increasing order. If the distribution lies completely in the bottom hemisphere, the first and last angles must be 0° and 90°, respectively. If the distribution lies completely in the top hemisphere, the first and last angles must be 90° and 180°, respectively. Otherwise, they must be 0° and 180°, respectively.

15 The set of horizontal angles, listed in increasing order. The first angle must be 0°. The last angle determines the degree of lateral symmetry displayed by the intensity distribution. If it is 0°, the distribution is axially symmetric. If it is 90°, the distribution is symmetric in each quadrant. If it is 180°, the distribution is symmetric about a vertical plane. If it is greater than 180° and less than or equal to 360°, the distribution exhibits no lateral symmetries. All other values are invalid.

16 The set of candela values. First, all the candela values corresponding to the first horizontal angle are listed, starting with the value corresponding to the smallest vertical angle and moving up the associated vertical plane. Then, the candela values corresponding to the vertical plane through the second horizontal angle are listed, and so on until the last horizontal angle. Each vertical slice of values must start on a new line. Long lines may be broken between values as needed by following the instructions given earlier.

Example of Photometric Data File

The following is an example of a photometric data file.

```
IESNA91
[TEST] Simple demo intensity distribution
[MANUFAC] Lightscape Technologies, Inc.
TILT=NONE
1
-1
1
8
1
1
2
0.0 0.0 0.0
1.0 1.0 0.0
0.0 5.0 10.0 20.0 30.0 45.0 65.0 90.0
0.0
1000.0 1100.0 1300.0 1150.0 930.0 650.0 350.0 0.0
```

Quick Reference

Commands

CONVERTOLDLIGHTS

Converts lights created in previous drawing file formats to the current format.

DISTANTLIGHT

Creates a distant light.

FREESPOT

Creates free spotlight which is similar to a spotlight without a specified target.

FREEWEB

Creates a free web light which is similar to a web light without a specified target.

GEOGRAPHICLOCATION

Specifies the geographic location information for a drawing file.

LIGHT

Creates a light.

LIGHTLIST

Turns on and off the Lights in Model palette that lists all lights in the model.

POINTLIGHT

Creates a point light that radiates light in all directions from its location.

RENDEREXPOSURE

Provides settings to adjust the global lighting for the most recently rendered output.

SPOTLIGHT

Creates a spotlight that emits a directional cone of light.

SUNPROPERTIES

Displays the Sun Properties window.

TARGETPOINT

Creates a target point light.

WEBLIGHT

Creates a web light.

System Variables

3DCONVERSIONMODE

Used to convert material and light definitions to the current product release.

DEFAULTLIGHTING

Turns on and off default lighting in place of other lighting.

DEFAULTLIGHTINGTYPE

Specifies the type of default lighting, old or new.

LIGHTGLYPHDISPLAY

Turns on and off the display of light glyphs.

LIGHTINGUNITS

Controls whether generic or photometric lights are used, and specifies the lighting units for the drawing.

LIGHTSINBLOCKS

Controls whether lights contained in blocks are used when rendering.

LINEARBRIGHTNESS

Controls the brightness level of the viewport when using default lighting or generic lights.

LINEARCONTRAST

Controls the contrast level of the viewport when using default lighting or generic lights.

LOGEXPBRIGHTNESS

Controls the brightness level of the viewport when using photometric lighting.

LOGEXPCONTRAST

Controls the contrast level of the viewport when using photometric lighting.

LOGEXPDAYLIGHT

Controls if the exterior daylight flag is enabled when using photometric lighting.

LOGEXPMIDTONES

Controls the mid tones level of the viewport when using photometric lighting.

RENDERUSERLIGHTS

Controls whether to override the setting for viewport lighting during rendering.

SUNSTATUS

Turns on and off the lighting effects of the sun in the current viewport.

Common Lamp Values for Photometric Lights

Lists of commonly used lamps for defining photometric lights are available on the Lighting tool palette.

Commonly used lamps values are available on the Lighting tool palette: Fluorescent; Low Pressure Sodium; Incandescent, and High Intensity Discharge.

Quick Reference

Commands

FREEWEB

Creates a free web light which is similar to a web light without a specified target.

LIGHT

Creates a light.

LIGHTLIST

Turns on and off the Lights in Model palette that lists all lights in the model.

RENDEREXPOSURE

Provides settings to adjust the global lighting for the most recently rendered output.

WEBLIGHT

Creates a web light.

System Variables

LIGHTGLYPHDISPLAY

Turns on and off the display of light glyphs.

LIGHTINGUNITS

Controls whether generic or photometric lights are used, and specifies the lighting units for the drawing.

LIGHTSINBLOCKS

Controls whether lights contained in blocks are used when rendering.

LINEARBRIGHTNESS

Controls the brightness level of the viewport when using default lighting or generic lights.

LINEARCONTRAST

Controls the contrast level of the viewport when using default lighting or generic lights.

LOGEXPBRIGHTNESS

Controls the brightness level of the viewport when using photometric lighting.

LOGEXPCONTRAST

Controls the contrast level of the viewport when using photometric lighting.

LOGEXPDAYLIGHT

Controls if the exterior daylight flag is enabled when using photometric lighting.

LOGEXPMIDTONES

Controls the mid tones level of the viewport when using photometric lighting.

RENDERUSERLIGHTS

Controls whether to override the setting for viewport lighting during rendering.

Use Distant Lights

Distant lights are useful for lighting objects or as a backdrop.

Distant Lights in Standard Lighting Workflow

A distant light emits uniform parallel light rays in one direction only. You specify a FROM point and a TO point anywhere in the viewport to define the direction of the light. Spotlights and point lights are each represented by a different light glyph. Distant lights are not represented by glyphs in the drawing because they do not have a discrete position and affect the entire scene.

The intensity of a distant light does not diminish over distance; it is as bright at each face it strikes as it is at the source. Distant lights are useful for lighting objects or for lighting a backdrop uniformly.

NOTE It is recommended that you do not use distant lights in blocks.

Distant Lights in Photometric Workflow

Distant lights are not physically accurate. It is recommended that you do not use them in a photometric workflow.

To create a distant light

1. Click Render tab ➤ Lights panel ➤ Lighting Units drop-down ➤ Generic Lighing Units. 

   No lighting units are used and standard (generic) lighting is enabled.

   NOTE Alternatively, you can enter **lightingunits** at the command prompt and set the value to 0 for standard (generic) lighting.

2. Click Render tab ➤ Lights panel ➤ Create Light drop-down ➤ Distant. 

3. Click in the drawing to specify a location for the light.
4. Click to specify a direction.
5. At the command prompt, enter **n** and enter a name. This name will appear in the properties and in the Lights in Model window (LIGHTLIST).

   You can continue to specify properties by entering options, or you can exit and set properties interactively. When you use the interactive method, you can see the results of your changes as you work.

6 Press Enter twice to exit the command. A distant light is not displayed as a light glyph.

To change the properties of a distant light. Enter lightlist at the command prompt. In the Lights in Model window, double click on the distant light name from the light list and use the Lighting Properties palette to change its color and other properties.

Quick Reference

Commands

DISTANTLIGHT

Creates a distant light.

System Variables

LIGHTINGUNITS

Controls whether generic or photometric lights are used, and specifies the lighting units for the drawing.

Assigning a Shape to a Light

Assigning a shape to a light modifies the illumination of a scene.

Area and Linear Lights

The *Area* parameter on the light is a property of a light. Just as a light can have a color, it can also be assigned a shape. For example, you can shape it like a rectangle so that it acts like panel lighting in a ceiling. Or you can shape it like a line, so it acts like a narrow fluorescent light tube. The area light is a way to assign a shape to the light. The shape affects the rendering and shadows in the same way that a panel light casts different light than a tube light in the real world.

The Shape Property

The Shape property is available in the Lighting Properties palette. Under the Render Shadow Detail panel, there is a Type property. Whether the Shape property is displayed depends on the Type property that is selected.

The following types are available: Soft (shadow) map, Sharp (default), Soft (sampled). By selecting the Soft (sampled) option, the Shape property becomes available.

The available shapes depend on the type of light. You can select the Type property for the light distribution under the General panel in the Lighting category. If Spotlight and Soft (sampled) are selected, then the Shape types available are Rectangular and Disk. If Point and Soft (sampled) are selected, then the following shapes are available: Linear, Rectangular, Disk, Cylinder, and Sphere. If Web and Soft (sampled) are selected, then the following shapes are available: Linear, Rectangular, Disk, Cylinder, and Sphere. You can use the samples property on the area light to control the trade-off between rendering time and shadow accuracy.

The Visible Render property found under the Rendered Shadows Details panel controls the visibility of the shape when the scene is rendered.

The Visible in Render option is also displayed under ShadowDetail and controls the visibility of the shape when the scene is rendered.

To create a point light in photometric workflow and assign a rectangular shape to the light

1. At the command prompt, enter **lightingunits** and set the value to 1 (American units) or 2 (International SI units) for photometric lighting.
2. Click Render tab ➤ Lights panel ➤ Create Light drop-down ➤ Point.



3. Click in the drawing to specify a location for the light.
4. At the Command prompt, enter **n** and enter a name. This name will appear in the properties and in the Lights in Model window (LIGHTLIST).

   You can continue to specify properties by entering options, or you can exit and set properties interactively. When you use the interactive method, you can see the results of your changes as you work.
5. Press Enter twice to exit the command.

   Select the light and use grip tools to change the light.
6. In the drawing, right-click the light and click Properties. Use the Lighting Properties palette to change its properties.
7. On the Lighting Properties palette, under the General panel, verify that the distribution type for the light is Point.

8 Scroll down the Lighting Properties palette to the Rendered Shadow details.

9 Change the Type property to Soft (sampled).

10 Change the Shape property to Rectangular. Add dimensions to the Length and Width properties.

11 Change the Visible Render property to Yes.

12 Render the image. The light is represented as a rectangle in the drawing.

Adjust and Manipulate Lights

You can add point lights, spotlights, and distant lights and set the location and properties of each.

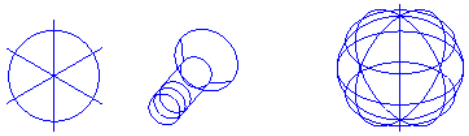
Control the Display of Lights

The display of lights can be turned on and off in the drawing.

A light glyph is a graphic representation of a light. Point lights and spotlights can be placed in a drawing with a light glyph. Distant lights, such as sunlight, are not represented with a light glyph.

The display of lights can be controlled several ways. In the Options dialog box, on the Drafting tab, the Lights Glyph settings control the display of light glyphs in the drawing.

Displaying light glyphs in the plotted drawing is optional; light glyph display is controlled with the plot glyph property setting. With the plot glyph property, you specify light glyph to display one light at a time. The plot glyph setting for a viewport affects all the lights globally.



To control the appearance of light glyphs

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Drafting tab, click Lights Glyph Settings.
3. In the Light Glyph Appearance dialog box, click Point or Spot to display a point light or spotlight glyph in Glyph Preview.
4. To change colors, click Edit Glyph Colors to open the Drawing Window Colors dialog box.

   For more information, see Set Interface Options on page 149
5. To change the size of the glyph in the drawing, use the Glyph Size slider.
6. Click OK to exit each dialog box.

To show or hide light glyphs

➤ Click Render tab ➤ Lights panel ➤ Light Glyph Display. 

Turns on and off the display of glyphs that represents lights in the current viewport.

NOTE The display setting does not control plotting of light glyphs.

To plot light glyphs

1. Click View tab ➤ Palettes panel ➤ Properties. 
2. Select a point light or a spotlight.
3. In the Properties palette, change the Plot Glyph property to **Yes**.

Quick Reference

Commands

DISTANTLIGHT

Creates a distant light.

LIGHT

Creates a light.

LIGHTLIST

Turns on and off the Lights in Model palette that lists all lights in the model.

LIGHTLISTCLOSE

Closes the Lights in Model window.

OPTIONS

Customizes the program settings.

POINTLIGHT

Creates a point light that radiates light in all directions from its location.

SPOTLIGHT

Creates a spotlight that emits a directional cone of light.

System Variables

LIGHTGLYPHDISPLAY

Turns on and off the display of light glyphs.

LIGHTLISTSTATE

Indicates whether the Lights in Model window is open or closed.

Adjust Light Placement

After a light has been placed in a scene the position and target can be modified.

The light, which is represented by a light glyph, can be repositioned after it is placed in the drawing. The light can be moved and rotated; the target can be modified. When the light glyph is selected, several grips are displayed.

By selecting the light with a right-click, a shortcut menu is displayed that provides the following options for adjusting the placement of a light:

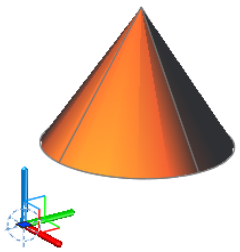
- **Move**. Activates the MOVE command. Used to change the position of the light.
- **Rotate.** Activates the ROTATE command. Used to change the orientation of lights. In addition to shortcut menu access, you can use ROTATE, 3DROTATE, and ROTATE3D commands.

- **Flip**. Rotates the target of the light in the opposite direction.

NOTE Rotating a targeted light is useful for aligning the area shadow region appropriately. Also, the orientation of the area shadow light is reset when the position or the target of the light is changed.

Location (Point Lights and Spotlights)

You can use the grip labeled Position to move a point light or a spotlight, or you can set the location in the Properties palette. The Position grip moves the light but does not change the target. To move both the light and its target, drag the light glyph itself.



To select a light using the light list

1. Click Render tab ➤ Lights panel ➤ Dialog box launcher. 
2. In the Lights in Model window, select the light name.

   If the light has a light glyph, the light displays as selected. By double clicking the name of the light, the Lighting Properties palette is displayed.

   NOTE Distant lights do not display a glyph in the drawing. Use the Lights in Model window to select a distant light.

To delete a light

1. Click Render tab ➤ Lights panel ➤ Dialog box launcher. 
2. Select one or more lights to delete.
3. Right-click the selection. Click Delete Light or Delete Lights.

To move a point light or a spotlight

1. Select the light glyph, or select the light in the Lights in Model window.
2. Select the Position grip.
   The Position grip is at the bottom of a spotlight and at the center of a point light.
3. Drag the light to the new location and click to place it there.

NOTE A spotlight rotates as it moves to maintain its aim at the target. If you want to move both source and target, drag the light glyph itself, not the Position grip.

Quick Reference

Commands

CONVERTOLDLIGHTS

Converts lights created in previous drawing file formats to the current format.

DISTANTLIGHT

Creates a distant light.

FREESPOT

Creates free spotlight which is similar to a spotlight without a specified target.

FREEWEB

Creates a free web light which is similar to a web light without a specified target.

GEOGRAPHICLOCATION

Specifies the geographic location information for a drawing file.

LIGHT

Creates a light.

LIGHTLIST

Turns on and off the Lights in Model palette that lists all lights in the model.

POINTLIGHT

Creates a point light that radiates light in all directions from its location.

RENDEREXPOSURE

Provides settings to adjust the global lighting for the most recently rendered output.

SPOTLIGHT

Creates a spotlight that emits a directional cone of light.

SUNPROPERTIES

Displays the Sun Properties window.

TARGETPOINT

Creates a target point light.

WEBLIGHT

Creates a web light.

System Variables

DEFAULTLIGHTING

Turns on and off default lighting in place of other lighting.

DEFAULTLIGHTINGTYPE

Specifies the type of default lighting, old or new.

LIGHTGLYPHDISPLAY

Turns on and off the display of light glyphs.

LIGHTINGUNITS

Controls whether generic or photometric lights are used, and specifies the lighting units for the drawing.

LIGHTSINBLOCKS

Controls whether lights contained in blocks are used when rendering.

LINEARBRIGHTNESS

Controls the brightness level of the viewport when using default lighting or generic lights.

LINEARCONTRAST

Controls the contrast level of the viewport when using default lighting or generic lights.

LOGEXPBRIGHTNESS

Controls the brightness level of the viewport when using photometric lighting.

LOGEXPCONTRAST

Controls the contrast level of the viewport when using photometric lighting.

LOGEXPDAYLIGHT

Controls if the exterior daylight flag is enabled when using photometric lighting.

LOGEXPMIDTONES

Controls the mid tones level of the viewport when using photometric lighting.

RENDERUSERLIGHTS

Controls whether to override the setting for viewport lighting during rendering.

SUNSTATUS

Turns on and off the lighting effects of the sun in the current viewport.

Control Light Properties

Every light in the drawing has general and specific lighting properties that can be changed after the light is placed.

Each light you add to the drawing is listed in the Lights in Model palette with the exception of lights in blocks and xrefs, the sun, and default lighting. You can selected a light either from the list or in the drawing. All properties can be changed in the Properties palette.

When a light is selected, its properties can be changed in the Properties Inspector.

You can use grip tools to move or rotate a selected light and change other properties such as the hotspot and falloff cone in spotlights. You can see the effect on the model as you change the properties of a light.

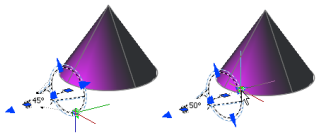
General Properties

The following properties are common to all lights. Full descriptions of the controls are located under the Properties command in Lighting Properties:

- **Name**. Specifies the name assigned to the light.
- **Type**. Specifies the type of light: point light, spotlight, distant light, or web.
- **On/Off Status**. Controls whether the light is turned on or off.
- **Shadows**. Controls whether the light casts shadows. To be displayed, shadows must be turned on in the visual style applied to the current viewport. Turn shadows off to increase performance.
- **Intensity factor**. Sets a multiplier that controls brightness. Intensity is not related to attenuation.
- **Filter color**. Sets the color of the light emitted.
- **Plot glyph**. Allows the ability to plot the drawing with the light glyphs on.

Spotlight Hotspot and Falloff Under General Properties

When light from a spotlight falls on a surface, the area of maximum illumination is surrounded by an area of lesser intensity.



- **Hotspot cone angle**. Defines the brightest part of a light beam. Also known as the *beam angle*.
- **Falloff cone angle**. Defines the full cone of light. Also known as the *field angle*.

- **Rapid decay area**. Consists of the region between the hotspot and falloff angles.

The greater the difference between the hotspot and falloff angles, the softer the edge of the light beam. If the hotspot and falloff angles are near equal, the edge of the light beam is sharp. Both values can range from 0 to 160 degrees. You can adjust these values directly with the Hotspot and Falloff grips.

Photometric Properties

Photometric lighting offers additional properties that make the lighting different than standard lighting. The following properties are under the Photometric properties panel:

- **Lamp intensity.** Specifies the inherent brightness of the light. Specifies the intensity, flux, or illuminance of the lamp.
- **Resulting intensity.** Gives the final brightness of the light. (Product of lamp intensity and intensity factor. Read-only.)
- **Lamp color.** Specifies the inherent color of the light in Kelvin temperature or standard.
- **Resulting color.** Gives the final color of the light. This is determined by a combination of the lamp color and the filter color. (Product of lamp color and filter color. Read-only.)

If you select Web in the Type property for a photometric light, additional properties are offered in the Photometric Web and Web offsets panel in the Lighting category.

- **Web file.** Specifies the data file describing the intensity distribution of the light.
- **Web preview.** Displays a 2D slice through goniometric data.
- **Rotation of X.** Specifies a rotational offset of the web about the optical X axis.
- **Rotation of Y.** Specifies a rotational offset of the web about the optical Y axis.
- **Rotation of Z.** Specifies a rotational offset of the web about the optical Z axis.

Geometry Properties

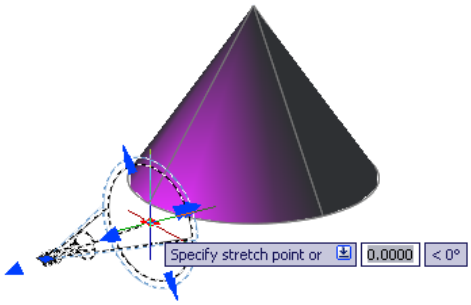
The Geometry properties control for the location and target point of the light. If the light is a target point light, spotlight, or weblight, additional target point properties are available. The Target property of a light can also be turned on an off.

Attenuation Properties (Point Lights and Spotlights)

Attenuation controls how light diminishes over distance. The farther away an object is from a light, the darker the object appears. You can specify no attenuation, inverse linear, or inverse squared (POINTLIGHT, SPOTLIGHT). Attenuation is not active for photometric lights.

- **None**. Sets no attenuation. Objects far from the point light are as bright as objects close to the light.
- **Inverse Linear**. Sets attenuation to be the inverse of the linear distance from the light. For example, at a distance of 2 units, light is half as strong as at the point light; at a distance of 4 units, light is one quarter as strong. The default value for inverse linear is half the maximum intensity.
- **Inverse Square**. Sets attenuation to be the inverse of the square of the distance from the light. For example, at a distance of 2 units, light is one quarter as strong as at the point light; at a distance of 4 units, light is one sixteenth as strong.

Another way to control the start point and end point of light is to use limits. Limits work like clipping planes to control where light is first emitted and where it stops. Using limits can increase performance by removing the need for the program to calculate light levels where the light is already practically invisible.



To set attenuation in a point light or a spotlight

1 Click Render tab ➤ Lights panel ➤ Lighting Units drop-down ➤ Generic Lighing Units. 

No lighting units are used and standard (generic) lighting is enabled.

NOTE Alternatively, you can enter **lightingunits** at the command prompt and set the value to 0 for standard (generic) lighting.

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2 Click View tab ➤ Palettes panel ➤ Properties. 

3 In the drawing, click the light glyph.

4 In the Properties palette, under Attenuation, set Use Limits to Yes, or select a type of attenuation.

5 If you set Use Limits to Yes, enter the following values in the properties palette or use the grip tool in the drawing to set the limits:

- **Start Limit Offset**. Specifies the point where light starts as an offset from the center of the light. The default is 0.
- **End Limit Offset**. Specifies the point where light ends as an offset from the center of the light. No light is cast beyond this point.

To specify a source vector for a distant light

1 Click Render tab ➤ Lights panel ➤ Lighting Units drop-down ➤ Generic Lighing Units. 

No lighting units are used and standard (generic) lighting is enabled.

NOTE Alternatively, you can enter **lightingunits** at the command prompt and set the value to 0 for standard (generic) lighting.

2 Click Render tab ➤ Lights panel ➤ Lights In Model. 

3 In the Lights in Model palette, select the distant light.

4 Click View tab ➤ Palettes panel ➤ Properties. 

5 In the Properties palette, under Geometry, use one of the following methods to change a value:

- Enter a value in a cell.
- Click the Calculator button to use QuickCalc.
- Click the Select in Drawing button and specify a From point and a To point with the pointing device. The line displayed in the drawing represents the vector.

When you change From Vector and To Vector values, the Source Vector values are updated. When you change Source Vector values directly, the From Vector and To Vector values are updated.

To change the color of a light

1 Click View tab ➤ Palettes panel ➤ Properties. 

2 In the drawing, select the light glyph.

3 In the Properties palette, click in the Color cell and select a color. Click Select Color to open the Select Color dialog box.

Quick Reference

Commands

CONVERTOLDLIGHTS

Converts lights created in previous drawing file formats to the current format.

DISTANTLIGHT

Creates a distant light.

FREESPOT

Creates free spotlight which is similar to a spotlight without a specified target.

FREEWEB

Creates a free web light which is similar to a web light without a specified target.

GEOGRAPHICLOCATION

Specifies the geographic location information for a drawing file.

LIGHT

Creates a light.

LIGHTLIST

Turns on and off the Lights in Model palette that lists all lights in the model.

POINTLIGHT

Creates a point light that radiates light in all directions from its location.

RENDEREXPOSURE

Provides settings to adjust the global lighting for the most recently rendered output.

SPOTLIGHT

Creates a spotlight that emits a directional cone of light.

SUNPROPERTIES

Displays the Sun Properties window.

TARGETPOINT

Creates a target point light.

WEBLIGHT

Creates a web light.

System Variables

DEFAULTLIGHTING

Turns on and off default lighting in place of other lighting.

DEFAULTLIGHTINGTYPE

Specifies the type of default lighting, old or new.

LIGHTGLYPHDISPLAY

Turns on and off the display of light glyphs.

LIGHTINGUNITS

Controls whether generic or photometric lights are used, and specifies the lighting units for the drawing.

LIGHTSINBLOCKS

Controls whether lights contained in blocks are used when rendering.

LINEARBRIGHTNESS

Controls the brightness level of the viewport when using default lighting or generic lights.

LINEARCONTRAST

Controls the contrast level of the viewport when using default lighting or generic lights.

LOGEXPBRIGHTNESS

Controls the brightness level of the viewport when using photometric lighting.

LOGEXPCONTRAST

Controls the contrast level of the viewport when using photometric lighting.

LOGEXPDAYLIGHT

Controls if the exterior daylight flag is enabled when using photometric lighting.

LOGEXPMIDTONES

Controls the mid tones level of the viewport when using photometric lighting.

RENDERUSERLIGHTS

Controls whether to override the setting for viewport lighting during rendering.

SUNSTATUS

Turns on and off the lighting effects of the sun in the current viewport.

Sun and Sky Simulation

The sun is a light that simulates the effect of sunlight and can be used to show how the shadows cast by a structure affect the surrounding area.

Sun and sky are the primary sources of natural illumination in AutoCAD. Whereas the rays of the sun are parallel and of a yellowish hue, the light cast from the atmosphere comes from all directions and is distinctly bluish in color. When the LIGHTINGUNITS system variable is set to photometric, more sun properties are available.

When the workflow is photometric (the LIGHTINGUNITS system variable is set to 1 or 2) the sun properties have more properties available and are rendered using a more physically accurate sunlight model. The sun color is disabled for the photometric sun; the color is computed automatically based on the time, date, and location specified in the drawing. The color is determined based on the position in the sky. When the workflow is generic or standard lighting (the LIGHTINGUNITS system variable is set to 0), the additional sun and sky properties are unavailable.

The properties of the sun can be modified by using the SUNPROPERTIES command. The Sun Properties window is displayed. The details of the settings are located under SUNPROPERTIES in the *Command Reference*.

The rays of the sun are parallel and have the same intensity at any distance. Shadows can be on or off. To improve performance, turn off shadows when you don't need them. All settings for the sun except geographic location are saved per viewport, not per drawing. Geographic location is saved per drawing.

The angle of the light from the sun is controlled by the geographic location you specify for your model and by the date and time of day. These are properties of the sun and can be changed in the Sun Properties window and the Geographic Location dialog box. The time zone used is based on the location, but you can adjust it independently (TIMEZONE system variable).

Sky Background

The option to choose the sky background is only available when the lighting unit is photometric (the LIGHTINGUNITS system variable is set to 1 or 2). If you choose the sky background and change the lighting to standard (generic) lighting (the LIGHTINGUNITS system variable is set to 0), the sky background is disabled.

The sun and sky background can be interactively adjusted in a view that can be activated in the Sun Properties window by clicking the first button in the title bar of the Sky Properties. This button activates the Adjust Sun & Sky Background dialog box. Here you can change properties and preview the changes to the background.

Sky Illumination

The view can have a sky background or a sky background and illumination. This option is in the Sky Properties Status drop-down list in the Sun Properties window. Additional sky illumination adds extra light to the scene, simulating the effect of light scattered by the atmosphere throughout the scene. This additional light is only visible in rendered output when final gather is enabled. This can be found in the Advanced Render Settings palette.

NOTE The default final gather setting of Auto enables final gather in rendering when sky illumination is turned on.

View

The VIEW command provides a mechanism for creating and managing named views. The sun and sky background can be assigned to a new named view under Background. The sky background relies on the sun data in the named view.

You can adjust sun and sky background interactively in a view in the Adjust Sun & Sky Background dialog box. Click the first button in the title bar of the Sky Properties. This button activates the Adjust Sun & Sky Background dialog box. Here you can change properties and preview the changes to the background.

To turn the sun on and off in a drawing

1 Click Render tab ➤ Lights panel ➤ Lighting Units drop-down ➤ Generic Lighing Units. 

No lighting units are used and standard (generic) lighting is enabled.

NOTE Alternatively, you can enter **lightingunits** at the command prompt and set the value to 0 for standard (generic) lighting.

2. Click Render tab ➤ Sun & Location panel ➤  ➤ Sun Properties.
3. In the General settings, click the Status setting, and select On or Off.

To change the brightness of the sun

1. Click Render tab ➤ Lights panel ➤ Lighting Units drop-down ➤ Generic Lighing Units. 

   No lighting units are used and standard (generic) lighting is enabled.

NOTE Alternatively, you can enter **lightingunits** at the command prompt and set the value to 0 for standard (generic) lighting.

2. Click Render tab ➤ Sun & Location panel ➤  ➤ Sun Properties.
3. In the General settings, click the Intensity Factor setting, and select a new value.

To change the geographic location of your model

1. Click Render tab ➤ Sun & Location panel ➤ Set Location. 
2. In the Location Already Exists task dialog box, click Edit Current Geographic Location.
3. In the Geographic Location dialog box, enter the appropriate values.
4. Click OK.

To change the angle of the sun

1. Click Render tab ➤ Sun & Location panel ➤  ➤ Sun Properties.
2. In the Sun Angle Calculator settings, click the Date setting, and select a new date.

3 Click the Time setting, and select a new time.
Use the Daylight Saving setting if necessary.

To change the color of the sun

1 Click Render tab ➤ Sun & Location panel ➤  ➤ Sun Properties.

2 In the General settings, click the Color setting, and select a color.
Click Select Color to open the Select Color dialog box.

To create a new view with the sun and sky background

1 At the command prompt, enter **lightingunits** and set the value to 1 (American units) or 2 (International SI units) for photometric lighting.

2 Click View tab ➤ Views panel ➤ Named Views. 

3 Click New.

4 In the New View dialog box, enter a name for the new view.

5 In the Background drop-down list, change the default to Sun & Sky.

6 In the Adjust Sun & Sky Background dialog box, make any adjustments to the background.

7 Click OK.

Quick Reference

Commands

GEOGRAPHICLOCATION

Specifies the geographic location information for a drawing file.

SUNPROPERTIES

Displays the Sun Properties window.

SUNPROPERTIESCLOSE

Closes the Sun Properties window.

System Variables

LATITUDE

Specifies the latitude of the drawing model in decimal format.

LONGITUDE

Specifies the longitude of the drawing model in decimal format.

NORTHDIRECTION

Specifies the angle of the sun from north.

SUNPROPERTIESSTATE

Indicates whether the Sun Properties window is open or closed.

SUNSTATUS

Turns on and off the lighting effects of the sun in the current viewport.

TIMEZONE

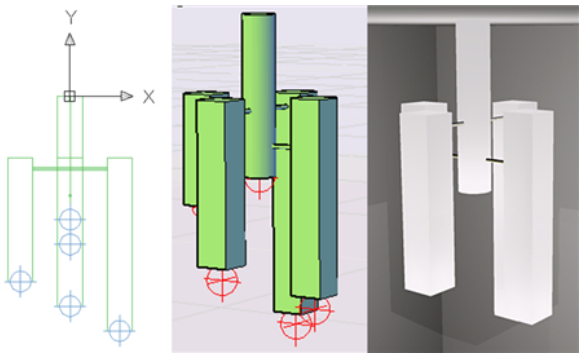
Sets the time zone for the sun in the drawing.

Incorporate Luminaire Objects

A luminaire object is a helper object that assembles a set of objects into a light fixture.

A luminaire object groups and manages the components of a light as a whole. Light fixtures can be represented by embedding photometric lights in blocks that also contain geometry. A luminaire object assembles a set of light objects into a light fixture.

Tool palettes of photometric lights are available to provide easy access to create a luminaire object. Luminaire objects are offered through plug-ins.



An example of a luminaire object.

To add a photometric light to be used in a luminaire

1. Click Render tab ➤ Lights panel ➤ .Lights drop-down ➤ At the command prompt, enter **lightingunits.**
2. Set the value to 1 (American lighting units) or 2 (International lighting units) for photometric lighting.
3. Click View tab ➤ Palettes panel ➤ Tool Palettes. 
4. Right-click the Tool palette title bar. Click Photometric Lights.
5. Drag a light from the palette to the drawing.
6. Drag another light from the palette to the drawing.
7. At the command prompt, enter **block** to create a light fixture with the lights placed in the drawing.

Quick Reference

System Variables

LIGHTSINBLOCKS

Controls whether lights contained in blocks are used when rendering.

Converting Lights

Properties of lights converting from previous releases of AutoCAD and other products.

Work with Lights in Drawings from Previous Versions of AutoCAD

Converting lights from previous versions.

With AutoCAD 2008, you can choose either standard (generic) lighting or photometric lighting. Prior to AutoCAD 2008, the only option was standard (generic) lighting. There is no explicit conversion required for lights from AutoCAD 2007 to AutoCAD 2008. Drawings from AutoCAD 2007 will open by default in AutoCAD 2008 in the standard (generic) lighting workflow. The additional photometric properties in AutoCAD 2008 are available as soon as you turn on the photometric lighting workflow in your drawing.

There is a conversion process required when opening drawings with lights from versions prior to AutoCAD 2007. The 3DCONVERSIONMODE system variable is set by default to convert drawings with lighting from previous versions to the AutoCAD 2007 and AutoCAD 2008 lighting format. The system variable has three settings. If the variable is set to 0, no automatic conversion takes place. If it is set to a value of 1, the default value, the conversion takes place automatically. If the value is set to 2, you are prompted when lights need to be converted and have the option to convert or not to convert.

You can use the CONVERTOLDLIGHTS command to convert the old lighting manually.

Quick Reference

Commands

CONVERTOLDLIGHTS

Converts lights created in previous drawing file formats to the current format.

System Variables

3DCONVERSIONMODE

Used to convert material and light definitions to the current product release.

Import Drawings into 3ds Max or VIZ

Drawing can be opened in 3ds Max or VIZ.

Please review the 3ds Max or VIZ DWG import documentation for information regarding the import of the DWG format.

Quick Reference

Commands

CONVERTOLDLIGHTS

Converts lights created in previous drawing file formats to the current format.

System Variables

3DCONVERSIONMODE

Used to convert material and light definitions to the current product release.

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Materials and Textures 45

Materials define the shininess, bumpiness, and transparency of object's surfaces to give them a realistic appearance.

Overview of Materials

Add materials to objects in your drawings to provide a realistic effect in any rendered view.

Autodesk provides a large library of predefined materials for you to use. Use the Materials Browser to browse materials and apply them to objects in your drawing.

Use the Materials Editor to create and modify materials.

Textures add complexity and realism to a material. For example, to replicate the bumps in a paved road, you could apply a Noise texture to an object representing a road in a drawing. To replicate a brick and mortar pattern, you could use a Tile texture. Use the Texture Editor to define a texture's appearance and how it is applied to an object.

Quick Reference

Commands

MATBROWSERCLOSE

Closes the Materials Browser.

MATBROWSEROPEN

Opens the Materials Browser.

MATEDITORCLOSE

Closes the Materials Editor.

MATEDITOROPEN

Opens the Materials Editor.

MATERIALATTACH

Associates materials with layers.

MATERIALMAP

Displays a material mapping gizmo to adjust the mapping on a face or an object.

MATERIALS

Opens the Materials Browser.

MATERIALSCLOSE

Closes the Materials Browser.

System Variables

3DCONVERSIONMODE

Used to convert material and light definitions to the current product release.

CMATERIAL

Sets the material of new objects.

MATBROWSERSTATE

Indicates whether the Materials Browser is open or closed.

MATEDITORSTATE

Indicates whether the Materials Editor is open or closed.

Create and Manage Materials

Access material libraries, or create and edit your own materials.

Browse Material Libraries

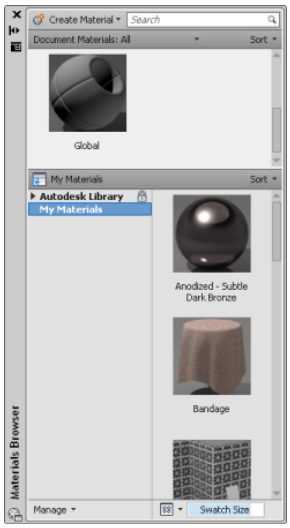
Browse, create, or open existing libraries from the Materials Browser.

Materials Browser

Use the Materials Browser to navigate and manage your materials. You can organize, sort, search, and select materials for use in your drawing.

The Autodesk library and user-defined libraries are accessible in the Materials Browser. The browser contains the following main components:

- **Browser toolbar.** Contains the Create Material menu, which allows you to create generic materials or from a list of templates, and the search box.
- **Document Materials.** Displays a set of display options for the materials saved in the current drawing. You can sort the document materials by name, type, and color.
- **Material library.** Displays the Autodesk library, which contains the predefined Autodesk materials and other libraries, which contain user-defined materials. It also contains a button for controlling the display of libraries and library categories. You can sort the materials in the library by name, category, type, and color.
- **Library details.** Displays previews of the materials in the selected categories.
- **Browser bottom bar.** Contains the Manage menu which provides options for adding, removing, and editing libraries and library categories. It also contains a button for controlling the display options for the library details.



Materials Libraries

The Autodesk library, with over 700 materials and over 1000 textures, is included with the product. The library is read-only, but you can copy Autodesk materials into the drawing, edit them, and save them to your own library.

There are three types of libraries:

- **Autodesk Library.** Contains predefined materials provided by Autodesk for use by all applications that support materials. It contains material-related resources, such as textures, thumbnails, and so on. You cannot edit the Autodesk library. All user-defined or modified materials are placed in user libraries.
- **User Library.** Contains all materials, except those in the Autodesk library, that you intend to share between drawings. You can copy, move, rename, or delete user libraries.
- **Embedded Library.** Contains a cache of materials used or defined in a drawing and available only to that drawing. This library is created automatically when the first Autodesk application using Autodesk materials is installed. You cannot rename this type of library. It is stored in the drawing.

Open a Library

You can access and open existing user libraries created locally or on a network and add them to your defined libraries in the Materials Browser. You can preview and select materials from multiple libraries.

The libraries are stored in a single document and can be shared with other users. However, any custom texture files used by the materials in a user library must be manually bundled with the user library.

Once connected to a library, you can browse and view its contents. Materials are categorized by type.

Create and Modify a Library

Add your materials to one of your libraries. To modify Autodesk materials, first copy them to your drawing.

Use the Manage drop-down list in the Materials Browser to add, rename, or remove libraries. You can also add categories and regroup library materials in the Materials Browser. Locked libraries cannot be edited and only unlocked libraries can be removed.

NOTE When you remove a library from the Materials Browser, the library file remains on the hard disk. You must manually delete the library file to reclaim hard disk space.

To access the Materials Browser

- Click Render tab ➤ Materials panel ➤ Materials Browser. 

To manage a library

1. Click Render tab ➤ Materials panel ➤ Materials Browser. 
2. In the Materials Browser, Manage drop-down, you can do the following:
   - **Open existing library.** Displays the Add Library dialog box (a standard file selection dialog box) where you can select any existing library.
   - **Create new library.** Displays the Create Library dialog box (a standard file selection dialog box) where you can save the new library.
   - **Remove library.** Select the library and click Remove Library.
   - **Create category.** Select the library and click Create Category.
   - **Delete category.** Select the category and click Delete Category.
   - **Rename.** Select the library or category and specify the new name.

To add a material to a tool palette

1. Click Render tab ➤ Materials panel ➤ Materials Browser. 
2. Click View tab ➤ Palettes panel ➤ Tool Palettes. 
3. Right-click the material you want to add.
4. Click Add To ➤ Active Tool Palette.

   A material tool is added the active tool palette, where it is easily accessible when you are working in a drawing.

NOTE If you change the properties of the material in the Materials Editor later, the material tool is not automatically updated. Delete the old material tool. Drag the updated swatch to create a new one.

Quick Reference

Commands

MATBROWSERCLOSE

Closes the Materials Browser.

MATBROWSEROPEN

Opens the Materials Browser.

MATEDITORCLOSE

Closes the Materials Editor.

MATEDITOROPEN

Opens the Materials Editor.

MATERIALATTACH

Associates materials with layers.

MATERIALMAP

Displays a material mapping gizmo to adjust the mapping on a face or an object.

MATERIALS

Opens the Materials Browser.

MATERIALSCLOSE

Closes the Materials Browser.

System Variables

3DCONVERSIONMODE

Used to convert material and light definitions to the current product release.

MATBROWSERSTATE

Indicates whether the Materials Browser is open or closed.

MATEDITORSTATE

Indicates whether the Materials Editor is open or closed.

Create New Materials

A material is defined by a number of properties. The available properties depend on the selected material type.

Create a new material in the Materials Browser or the Materials Editor. Select the type of material to create, or duplicate and modify an existing material.

After you set properties, you can further modify materials even more by using maps, such as texture or procedural maps.

In the Materials Editor, define the following properties:

- **Appearance.** Defines the material's appearance. The appearance properties are unique to each material. Use the Texture Editor to edit texture or procedural maps assigned to the material. You can also rename the material.
- **Information.** Defines or displays the keywords and descriptions associated with a given material. Some information is read-only.

One material is always available in a new drawing, GLOBAL. This material is applied to all objects by default until another is applied.

Materials Editor

The material type, Generic, has the following properties to refine your material.

Color

The color of a material on an object is different in different areas of the object. For example, when you look at a red sphere, it does not appear to be uniformly red. The sides away from the light appear to be a darker red than the sides facing the light. The reflection highlight appears the lightest red. In fact, if the red Sphere is very shiny, its highlight may appear to be white.

You can assign a color or a custom texture which can be either an image or a procedural texture to the material.

Image

Controls the base diffuse color map of the material. The diffuse color is the color that an object reflects when illuminated by direct daylight or artificial light.

Image Fade

Controls the composite between the base color and the diffuse image. The image fade property is only editable if an image is used.

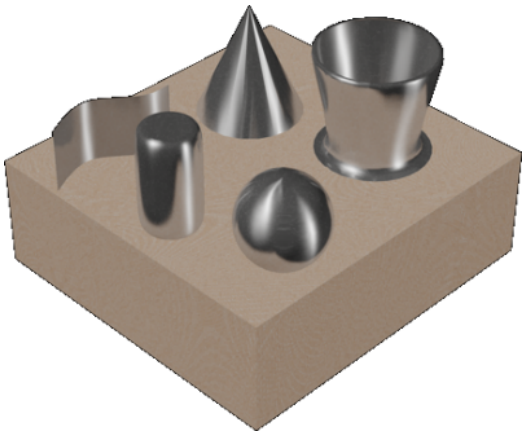
Glossiness

The reflective quality of the material defines the degree of glossiness or dullness. To simulate a glossy surface, the material has a small highlight, and its specular color is lighter, perhaps even white. A duller material has a larger highlight that is closer to the main color of the material.

Highlights

This controls the means for deriving specular highlights of the material.

Metallic highlights disperses light in an anisotropic way. *Anisotropic* refers to properties of material which depends on the direction. Metallic highlights are the color of the material while non-metallic are the color of the lights hitting the material.



Other Properties

The following properties can be used to create specific effects:

- **Reflectivity.** The Direct and Oblique sliders control the level of reflections and the intensity of the specular highlight on surfaces.
- **Transparency.** The Transparency check box controls the level of transparency of the material. A completely transparent object allows the passage of light through it. The transparency value is a percentage: at 1.0,

the material is completely transparent; at lower values, the material is partly opaque; and at 0.0, the material is completely opaque.
The Translucency and Index of Refraction properties becomes editable only when the Transparency value is greater than 0. A translucent object lets some light pass through and scatters some light within the object; for example, frosted glass. The translucency value is a percentage: at 0.0, the material is not translucent; at 1.0, the material is as translucent as possible.

The Index of Refraction controls the degree to which light rays are bent as they pass through the material and thus distort objects that are seen on the other side of the object. For example, at 1.0, the object behind the transparent object is not distorted. At 1.5, the object is distorted greatly, as if it were seen through a glass marble.

Material	Index of Refraction
Air	1.00
Water	1.33
Alcohol	1.36
Quartz	1.46
Glass	1.52
Diamond	2.30
Custom	0.00 - 5.00

- **Cutouts.** The Cutouts check box controls the perforation effects of the material based on a grayscale interpretation of a texture. Lighter areas of of the map render as opaque and darker areas render as transparent.
- **Self illumination.** The object appears to be emitting its own light. For example, to simulate neon without using a light source, you could set a self illumination value greater than zero. No light is cast on other objects. The Self Illumination check box infers changing values. This control filters color, luminance, and color temperature of the material. The Filter Color creates the effect of a color filter over the illuminated surface.

Luminance causes a material to simulate being lit within a photometric light source. How much light is emitted is a selected value in photometric units. No light is cast on other objects.

Material	Luminance (candelas per square meter)
Dim Glow	10
LED Panel	100
LED Screen	140
Cell Phone Screen	200
CRT Television	250
Lamp Shade Exterior	1300
Lamp Shade Interior	2500
Desk Lamp Lens	10000
Halogen Lamp Lens	10000
Frosted Bulb	210000
Custom	unclamped

The color temperature sets the color of the self illumination.

Material	Color Temperature (degrees Kelvin)
Candle	1850
Incandescent Bulb	2800
Flood Light	3400
Moonlight	4100
Daylight Warm	5000
Daylight Cool	6000

Material	Color Temperature (degrees Kelvin)
Xenon Arc Lamp	6420
TV Screen	9320
Custom	unclamped

- **Bump.** The Bump check box turns on or off the use of the relief patterns of the material. The object appears to have a bumpy or irregular surface. When you render an object with a bump-mapped material, the lighter areas of the map appear to be raised and the darker areas appear to be low. The Amount adjusts the height of the bump. Higher values render as higher relief and low values render as low relief. Grayscale images make effective bump maps.

To create a new material

1. Click Render tab ➤ Materials panel ➤ Materials Browser. 
2. In the Materials Browser, browser toolbar, click Create Material.
3. Select a material template.
4. In the Materials Editor, enter a name.
5. Specify the material color options.
6. Use the sliders to set the properties for the shininess, opacity, refraction, translucency, and so on.
7. (Optional) Select a map channel and a procedural map type. (See Use Map Channels for Added Texture Realism on page 2222 and Understand Map Types on page 2224.)

Quick Reference

Commands

CONVERTOLDMATERIALS

Converts materials created in previous drawing file formats to the current format.

MATBROWSERCLOSE

Closes the Materials Browser.

MATBROWSEROPEN

Opens the Materials Browser.

MATEDITORCLOSE

Closes the Materials Editor.

MATEDITOROPEN

Opens the Materials Editor.

MATERIALASSIGN

Uses the current material defined in the CMATERIAL system variable.

MATERIALATTACH

Associates materials with layers.

MATERIALMAP

Displays a material mapping gizmo to adjust the mapping on a face or an object.

MATERIALS

Opens the Materials Browser.

MATERIALSCLOSE

Closes the Materials Browser.

MIGRATEMATERIALS

Finds any legacy materials in Tool Palettes and converts them to generic type.

System Variables

3DCONVERSIONMODE

Used to convert material and light definitions to the current product release.

CMATERIAL

Sets the material of new objects.

LOGEXPPHYSICALSCALE

Controls the relative brightness of self-illuminated materials in a photometric environment.

MATBROWSERSTATE

Indicates whether the Materials Browser is open or closed.

MATEDITORSTATE

Indicates whether the Materials Editor is open or closed.

VSMATERIALMODE

Controls the display of materials in the current viewport.

Manage and Organize Materials

Duplicate materials to different libraries to create your own organization.

When a material is moved, a copy is created and it is added to the new category. If the material is copied into the root node, its original category is maintained and recreated in the new library. There are two ways to move materials:

- **Drag and drop.** You can drag swatches or materials from a library to the Materials in This Document section in the Materials Browser. You can also drag materials from a library to another library. A new copy of the material is created and saved with the drawing.
- **Shortcut menu.** You can use the shortcut menu to copy a material to a new library. Use the Add To option and then select, for example, material in the library copied to the document or a library material copied to another library.

Any in-document and unlocked material can be renamed in-place using the materials shortcut menu.

You can search the materials in all open libraries based on the material's name, description, and keyword information entered in the Materials Editor. All the materials are filtered to show only those matching the search string. Only the materials that have a match for the search string are displayed. Click the X button in the search box to clear the search and return to viewing the unfiltered library.

The search results depend on the library you have selected in the tree view. For example, if you select the Library root node, it displays the search results

for all matching materials in the selected library. But if you select a category, it searches only within that category.

You can delete a selected and unlocked material by using the shortcut menu or with the Delete key. Locked materials cannot be deleted from the Materials Browser or from the shortcut menu.

To add a material to a library

1. Click Render tab ➤ Materials panel ➤ Materials Browser. 
2. In the Materials Browser, right-click the material and click Add To.
3. Select the library where you want the material to be added. You can add the material to multiple libraries.

To search the objects using a material

1. Click Render tab ➤ Materials panel ➤ Materials Browser. 
2. In the Materials Browser, Document Materials, right-click a material and click Select Objects Applied To.

   Objects using the material are highlighted in the drawing.

To search for a material in the library

1. Click Render tab ➤ Materials panel ➤ Materials Browser. 
2. In the Materials Browser, browser toolbar, type the name of the material.

   Library nodes containing the materials matching the entered characters are displayed.

Quick Reference

Commands

MATBROWSERCLOSE

Closes the Materials Browser.

MATBROWSEROPEN

Opens the Materials Browser.

MATEDITORCLOSE

Closes the Materials Editor.

MATEDITOROPEN

Opens the Materials Editor.

MATERIALATTACH

Associates materials with layers.

MATERIALMAP

Displays a material mapping gizmo to adjust the mapping on a face or an object.

MATERIALS

Opens the Materials Browser.

MATERIALSCLOSE

Closes the Materials Browser.

System Variables

3DCONVERSIONMODE

Used to convert material and light definitions to the current product release.

MATBROWSERSTATE

Indicates whether the Materials Browser is open or closed.

MATEDITORSTATE

Indicates whether the Materials Editor is open or closed.

Modify Materials

After a material has been added to the drawing, you can modify it in the Materials Editor.

The material swatches that are available in the drawing are displayed in the Materials in This Document section in the Materials Browser. When you double-click a material swatch, the properties for the material become active in the Materials Editor sections.

As you modify the settings, they are saved with the material. The changes are displayed in the material swatch preview. By holding down the button below the swatch preview window, a set of flyout buttons display different geometry options for the material preview.

To change the name of a material

1. Click Render tab ➤ Materials panel ➤ Materials Browser. 
2. In the Materials Browser, select the material and right-click. Click Rename.
3. Change the name and description of the material in place.

To change the material preview shape and render quality

1. Click Render tab ➤ Materials panel ➤  ➤ Materials Editor. 
2. In the Materials Editor, click Options.
3. Select a different swatch shape and render quality for the material.

Quick Reference

Commands

MATBROWSERCLOSE

Closes the Materials Browser.

MATBROWSEROPEN

Opens the Materials Browser.

MATEDITORCLOSE

Closes the Materials Editor.

MATEDITOROPEN

Opens the Materials Editor.

MATERIALATTACH

Associates materials with layers.

MATERIALMAP

Displays a material mapping gizmo to adjust the mapping on a face or an object.

MATERIALS

Opens the Materials Browser.

MATERIALSCLOSE

Closes the Materials Browser.

System Variables

3DCONVERSIONMODE

Used to convert material and light definitions to the current product release.

MATBROWSERSTATE

Indicates whether the Materials Browser is open or closed.

MATEDITORSTATE

Indicates whether the Materials Editor is open or closed.

Convert Materials

Upgrade legacy materials from older files to the new format.

When a *legacy material*, any material created in AutoCAD 2010 or earlier releases, is shown in the Materials Editor, you have the option to convert the material into an editable format. After the conversion, the Materials Editor controls are enabled and editable.

Legacy materials are rendered but not editable in the application. It is recommended to convert these materials to their Generic equivalent. The new materials provide better results and can be shared across Autodesk applications.

Apply Materials to Objects and Faces

You can apply materials to objects, layers, or faces.

To apply a material to an object or a face (a triangular or quadrilateral portion of a surface object), first select the object and select the material from the Materials Browser. The material is added to the drawing, and it is also displayed as a swatch in the Materials Browser.

When you create or modify a material in the Materials Browser, you can

- Click a material in the library and the material is applied to any selected objects in your drawing.
- Drag the material swatch directly onto objects in your drawing.
- Assign a material to an object by clicking the Assign to Selection in the shortcut menu on the material swatch in the Materials Browser.

NOTE AutoCAD 2012 supports per-face color for all materials and textures. If an object has per-face color and a diffuse (color) texture, the per-face colors are shown when the texture is faded.

To apply a material to an object from a tool palette

1. Click View tab ➤ Palettes panel ➤ Tool Palettes. 
2. Click a tool palette tab to make it active.
3. Drag the material from the tool palette onto the object.

To apply a material to an object from the Materials Browser

1. Click Render tab ➤ Materials panel ➤ Materials Browser. 
2. Right-click the material you want to apply.
3. Click Assign to Selection.
4. Select the object.
   The material is applied to each object as you select it.

To apply a material by layer

1. At the Command prompt, enter **materialattach**. 
2. In the Material Attachment Options dialog box, drag a material onto a layer.

The material is applied to all objects on the layer whose Material property is set to BYLAYER. BYLAYER is the default for the Material property when you create an object.

Quick Reference

Commands

MATBROWSERCLOSE

Closes the Materials Browser.

MATBROWSEROPEN

Opens the Materials Browser.

MATEDITORCLOSE

Closes the Materials Editor.

MATEDITOROPEN

Opens the Materials Editor.

MATERIALATTACH

Associates materials with layers.

MATERIALS

Opens the Materials Browser.

MATERIALSCLOSE

Closes the Materials Browser.

System Variables

MATBROWSERSTATE

Indicates whether the Materials Browser is open or closed.

MATEDITORSTATE

Indicates whether the Materials Editor is open or closed.

Use Maps for Added Realism

Mapping is a way of increasing the visual complexity of a material.

Overview of Maps

The images you assign to materials are called *maps*. You use maps to improve the appearance and realism of materials. Materials that contain one or more images are called *mapped materials*.

Maps can simulate textures, reflections, refractions, and other effects. When used with materials, maps add details without adding complexity to the geometry of an object.

Use Map Channels for Added Texture Realism

Textures add the realism of details to a material.

Overview of Map Channels

Map channels assign a texture to a material's color. The colors of the map replace the material's diffuse color when a map channel is selected. For transparency, the texture is a multiplier.

After selecting a map channel you can use any image map, or one of the procedural maps, such as wood and marble. For example, if you want a wall to be made out of brick, you can choose a texture map with an image of bricks. This is the most common kind of mapping. You can also use a procedural map, such as tiles or wood. The procedural maps have properties you can adjust for the effect you want; for example, the tiling size and mortar spacing for a brick patterned material or the spacing of the grain in a wood material.

The map you add to a material can be scaled. The pattern can be tiled or rotated.

Maps are also available for other purposes. You can use more than one map for the same material and can create nested maps.

After you apply the map, you can adjust it to fit on the face or object by using material mapping.

NOTE When Texture Compression is turned on, the amount of video memory required to open a drawing that contains materials with images is decreased. By using texture compression, the amount of video memory necessary to display the drawing is reduced, but the time it takes to load the images the first time that they are accessed is increased. Also, there is a reduction in the quality of the images when they are displayed in the viewport or plotted. When you create a rendering, there is no loss in image quality.

Reflectivity Map

Reflectivity maps simulate a scene reflected on the surface of a shiny object.

Use a reflectivity map to simulate a scene reflected on the surface of a shiny object. For reflectivity maps to render well, the material should be shiny, and the reflection image itself should have a high resolution (at least 512 by 480 pixels).

Transparency Map

Transparency maps create the effect of transparency and opacity.

You can specify areas of transparency and opacity by using the transparency map channel. The effect of transparency is best previewed against a pattern background.

Cutouts Map

Cutouts maps make the material partially transparent.

You can select an image file to use for cutouts mapping. Transparent areas are fully transparent.



When you use transparency for frosting or translucent effects, reflectivity is maintained. Use a cutout in addition to make the material take a particular shape or pattern. Cutout areas do not reflect.

Self Illumination Map

Self illumination maps make portions of an object appear to glow.

White areas of the map render as fully self illuminating. Black areas render with no self illumination. Gray areas render as partially self illuminating, depending on the grayscale value. Self illumination means that the glowing area is not affected by lights in the drawing (its ambient color component goes away), and does not receive shadows.

Bump Map

A bump map simulates a bumpy or irregular surface.

You can select an image file or procedural maps to use for mapping. Bump mapping makes an object appear to have a bumpy or irregular surface. When you render an object with a bump-mapped material, lighter (whiter) areas of the map appear to be raised and darker (blacker) areas appear to be low. If the image is in color, the gray-scale value of each color is used. Bump mapping increases rendering time significantly but adds to the realism.

Use bump maps when you want to take the smoothness off a surface, or to create an embossed look. Keep in mind, however, that the depth effect of a bump map is limited because it does not affect the profile of the object and cannot be self-shadowing. If you want extreme depth in a surface, you should use modeling techniques instead. The bumps are a simulation created by perturbing face normals before the object is rendered. Because of this, bumps do not appear on the silhouette of bump-mapped objects.

The Bump map slider adjusts the degree of bumpiness. Higher values render as higher relief and negative values inverses the relief.

Understand Map Types

You can select texture or procedural maps within each map channel (Reflectivity, Transparency, Cutouts, Self Illumination, and Bump) to increase the complexity of a material.

Overview of Map Types

Each map type has its own specific set of map controls. Within each map channel, you can control settings, such as disable a map, delete a map, or synchronize a map.The Nested Map Navigation display is quick reference for the nested maps assigned to a material. To display the nested maps, click the drop-down arrow next to the name of the material. After you apply a map type, you can adjust it to fit on a face or object using material mapping.

Image Map

You can select texture or procedural maps within each map channel (Reflectivity, Transparency, Cutouts, Self Illumination, and Bump) to increase the complexity of a material.



You can use the following file types to create texture maps:

- BMP, RLE, or DIB
- GIF
- JFIF, JPG, or JPEG
- PCX

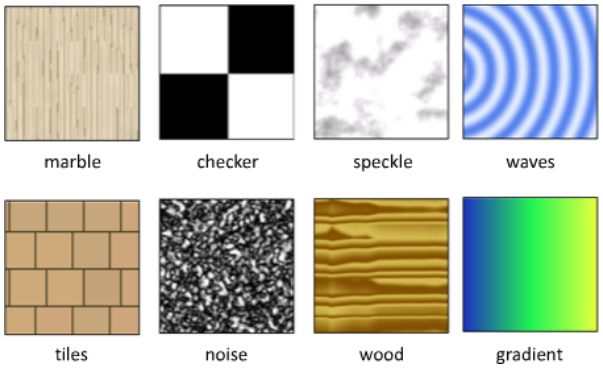
- PNG
- TGA
- TIFF

Procedural Maps

Procedural maps add further realism to a material.

Overview of Procedural Maps

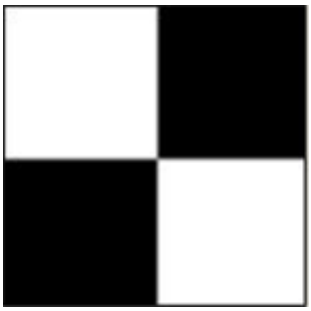
Unlike bitmap images, which are produced by a fixed matrix of colored pixels, a procedural map is generated by a mathematical algorithm. Consequently, the types of controls you find for a procedural map vary depending on the capabilities of the procedure. A procedural map can be generated in either two or three dimensions. You can also nest texture or procedural maps within another procedural map to add depth and complexity to the material.



Checker

Applies a two-color checkerboard pattern to the material.

The default checker map is a pattern of black and white squares. The component checks can be either colors or maps. You can preview this map in the swatch preview.



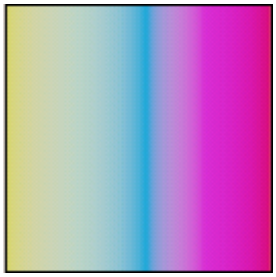
See also:

- Checker Property Settings

Gradient

Creates gradients using colors, and blends.

You can make highly customized gradients with the Gradient procedural map. Gradient uses several colors to create shades or ramps from one to another.



See also:

- Gradient Property Settings

Marble

Applies a stone and vein color pattern.

You can use the Marble map to specify stone and vein color. You can modify the vein spacing and vein width.



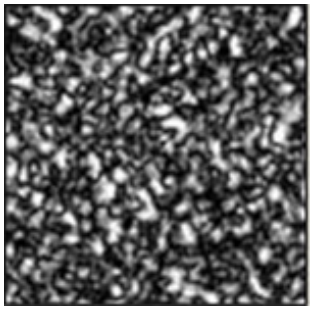
See also:

- Marble Property Settings

Noise

Creates random perturbation of a surface based on the interaction of two colors, texture maps or a combination.

You can use Noise to attenuate the repetitiveness aspect of bitmap and tiles. The noise procedural map uses two colors, sub-procedural maps or a combination of both to create a random pattern.



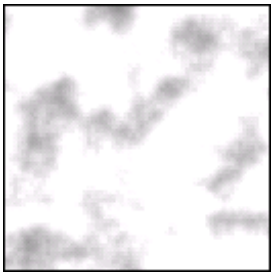
See also:

- Noise Property Settings

Speckle

Generates a speckled surface pattern.

The Speckle map is useful for diffuse mapping and bump mapping to create granite-like and other patterned surfaces.



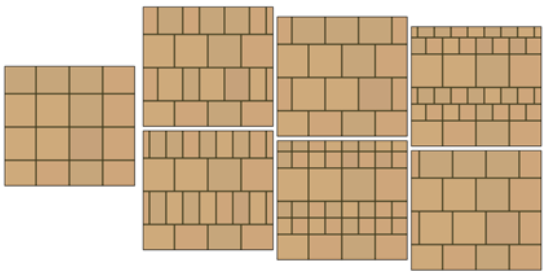
See also:

- Speckle Property Settings

Tiles

Applies a brick or stacked tiling of colors or material mappings.

You can apply an image and repeat the image as a pattern using Tiles. The Materials Browser provides commonly defined architectural brick patterns, which you can select and modify in the Materials Editor.



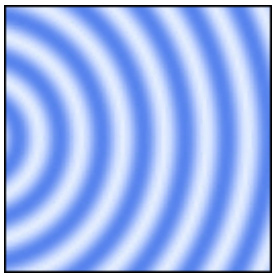
See also:

- Tiles Property Settings

Waves

Simulates water or wave effects.

You can use the Bump map to simulate the surface of a body of water. The Waves map generates a number of spherical wave centers and randomly distributes them over a sphere. You can control the number of wave sets, the amplitude, and the speed of the waves. This map works effectively as both a diffuse and bump map at the same time, or combined with an opacity map.



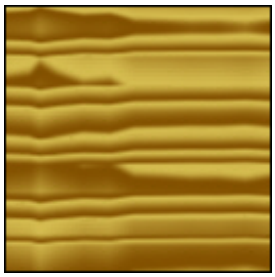
See also:

- Waves Property Settings

Wood

Creates the color and grain pattern of wood.

Use the Wood map to create the realistic color and grain properties of wood.



See also:

- Wood Property Settings

Modify Map Properties

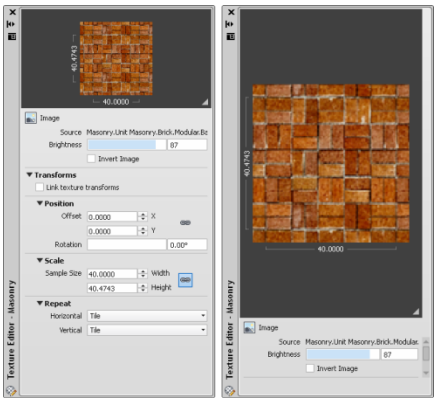
Map properties can be modified to create complex patterns by changing the map settings.

The map property settings displayed on the Texture Editor depend on the selected map type.

The Texture Editor, which is a modeless inspector panel, allows you to modify texture properties. This is displayed when you click on a texture swatch for a procedural or image-based texture.

Texture Preview

A preview of the texture swatch is displayed and any changes to the settings is reflected in the preview. You can enlarge or reduce the preview of the image in order to accurately manipulate the texture.

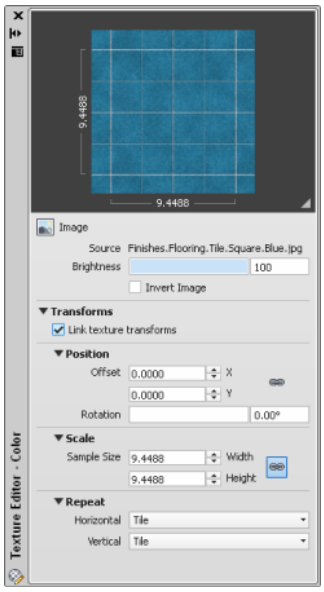


Appearance

The Appearance settings are only available for procedural-based textures.

Transformations

Linkable attributes are specific attributes of the same type that have been identified in the content design resulting from synchronization. Rotation is an example of a linkable attribute.When the Link Texture Transforms is enabled, all edits to all linkable attributes between all texture sockets are published. If it is disabled, the edits affect only the specific texture socket where the edit is made.Linked properties, such as Offset: X,Y and Sample Size: Width, Height, are not related to the concept of linkable attributes.



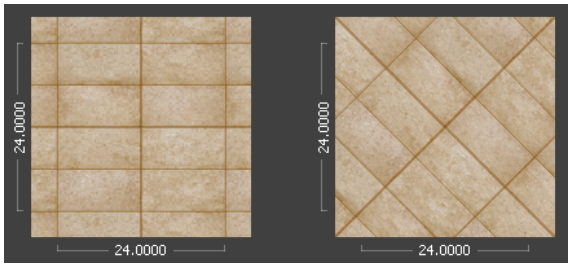
The Position, Scale, and Repeat are three standard transformations used in various texture sockets. The following identifies the type of texture (image and procedurals) where linkable attributes exist:

Texture Type	Position	Scale	Repeat
Image	Yes	Yes	Yes
Tiles	Yes	Yes	Yes
Gradient	Yes	Yes	Yes
Checker	Yes	Yes	Yes
Noise	Yes	No	No
Waves	Yes	No	No
Wood	Yes	No	No
Speckle	Yes	No	No

Texture Type	Position	Scale	Repeat
Marble	Yes	No	No

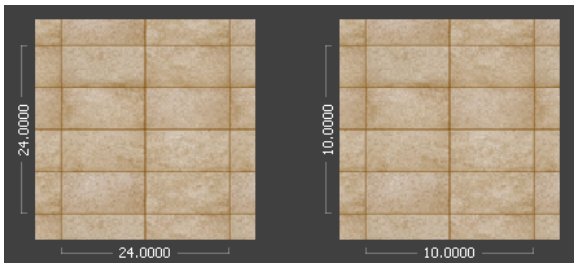
- **Position.** Each map has its own material offset and rotation factors. You can control the coordinates of the material on a swatch. You can also rotate the image around the W axis of the UVW coordinate system. As these settings are changed, a preview is displayed in the Texture Editor.

  NOTE UVW stands for the three axes of the texture space, similar to XYZ in world, object, camera, and other coordinate systems. Because of the way textures use the UV coordinates to project a bitmap on to a surface, coordinates with values above 1 cause the texture to repeat (or "tile"), virtually warping back to the same coordinate space from 0,0,0 to 1,1,1. This does not mean that UV coordinates have to be limited to values between 0 and 1. They can contain arbitrary values, both positive and negative.



rotated map

- **Scale.** You can specify real-world units to use in scaling.



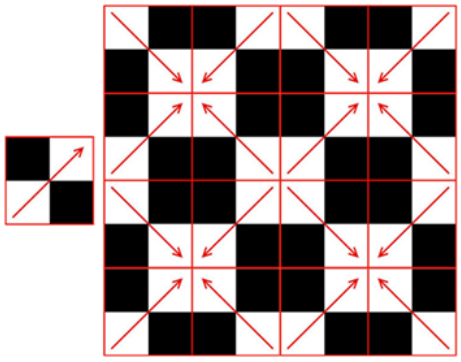
scaled map

- **Repeat.** You can tile or mirror a material to create a pattern, or select None for no modification to the map pattern.
  You can apply an image and repeat the image as a pattern using Tile. This effect is used to represent a tiled floor or fountain.



repeated map

Tiling wraps the selected object with the map image. In default mapping, tiling is active. Because the map is scaled to fit the object, you must offset the UV coordinates or rotate the map to see the tiling effect. The portions of the surface from which the image has moved away are filled by other portions of the map.



Mirrored Map

Mirroring doubles the map and flips the doubled copy. As with tiling, you can mirror in the U dimension, the V dimension, or both.

Adjust Mapping On Objects and Faces

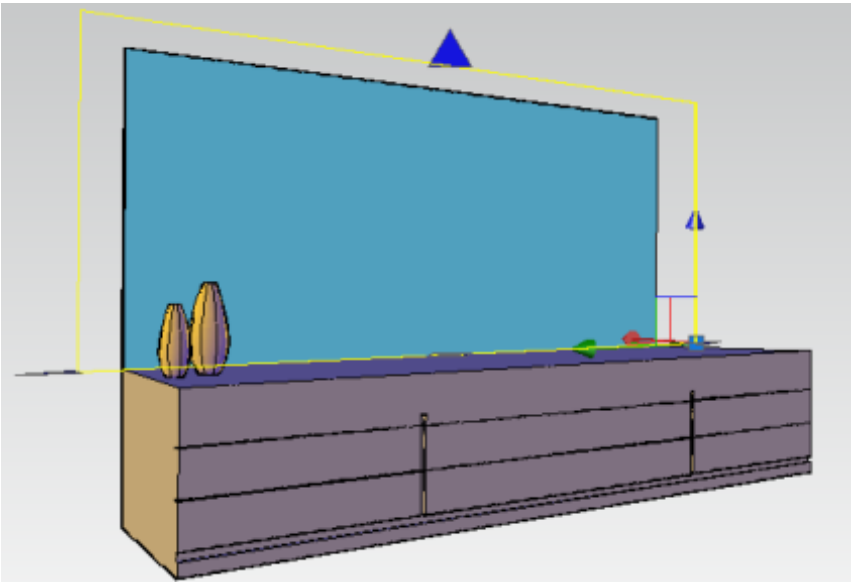
After you apply a texture to a material, you can adjust the orientation of the texture map on objects or faces.

When materials are mapped, you can adjust the material to fit the shape of the object. Applying the appropriate type of material mapping to an object improves the fit.

- **Planar mapping.** Maps the image onto the object as if you were projecting it from a slide projector onto a 2D surface. The image is not distorted from the direction of projection but is distorted if projected onto a curved surface and viewed at from the side. The image is not scaled to the object. This mapping is most commonly used for faces.
- **Box mapping.** Maps an image onto a boxlike solid. The image is repeated on each side of the object.
- **Spherical mapping.** Maps an image onto a spherical object. The top edge of the map is compressed to a point at the "north pole" of the sphere, as is the bottom edge at the "south pole."
- **Cylindrical mapping.** Maps an image onto a cylindrical object; the horizontal edges are wrapped together but not the top and bottom edges. The height of the image is scaled along the cylinder's axis.

Mapping Gizmo

If you need to make further adjustments, you can use the mapping gizmo that is displayed on the object to move or rotate the map on the object.



The mapping gizmos are viewport tools that let you quickly choose one or two axes when transforming a selection with the mouse. You choose an axis by placing the mouse over any axis of the icon, then drag the mouse to transform the selection along that axis. When moving or scaling an object, you can use other areas of the gizmo to perform transforms along any two axes simultaneously. Using a gizmo lets you switch quickly and easily between different transform axes and planes.

Quick Reference

Commands

MATBROWSERCLOSE

Closes the Materials Browser.

MATBROWSEROPEN

Opens the Materials Browser.

MATEDITORCLOSE

Closes the Materials Editor.

MATEDITOROPEN

Opens the Materials Editor.

MATERIALMAP

Displays a material mapping gizmo to adjust the mapping on a face or an object.

MATERIALS

Opens the Materials Browser.

MATERIALSCLOSE

Closes the Materials Browser.

System Variables

MATBROWSERSTATE

Indicates whether the Materials Browser is open or closed.

MATEDITORSTATE

Indicates whether the Materials Editor is open or closed.

Render 3D Objects for Realism

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A realistic rendering of a model can often give a product team or prospective client a clearer vision of a conceptual design than a plotted drawing.

Overview of Rendering

Rendering creates a 2D image based on your 3D scene. It shades the scene's geometry using the lighting you've set up, the materials you've applied, and environmental settings such as background and fog.



The renderer is a general-purpose renderer that generates physically correct simulations of lighting effects, including ray-traced reflections and refractions, and global illumination.

A range of standard rendering presets, reusable rendering parameters, are available. Some of the presets are tailored for relatively quick preview renderings while others are for higher quality renderings.

Quick Reference

Commands

RENDER

Creates a photorealistic or realistically shaded image of a 3D solid or surface model.

RENDERCROP

Renders a specified rectangular area, called a crop window, within a viewport.

RENDERENVIRONMENT

Controls visual cues for the apparent distance of objects.

RENDERPRESETS

Specifies render presets, reusable rendering parameters, for rendering an image.

RENDERWIN

Displays the Render window without starting a rendering operation.

RPREF

Displays or hides the Advanced Render Settings palette for access to advanced rendering settings.

RPREFCLOSE

Closes the Render Settings palette.

SAVEIMG

Saves a rendered image to a file.

Prepare a Model for Rendering

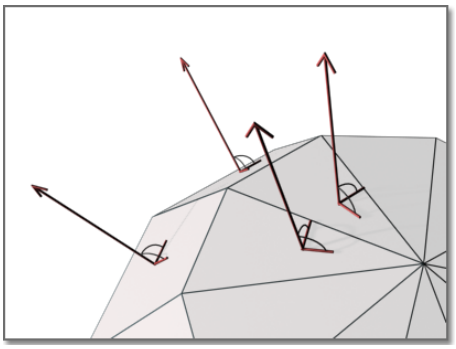
The way a model is built plays an important role in optimizing rendering performance and image quality.

Understand Face Normals and Hidden Surfaces

There are several steps commonly taken to speed up the rendering process.

In order to minimize the time it takes to render a model, it is common practice to remove hidden surfaces or hide objects that are positioned off-camera. Furthermore, ensuring that all face normals orient in the same direction can also speed up the rendering process.

Every surface that you model is made up of faces. Faces are either triangular or quadrilateral and each face has an inward and outward oriented side. The direction in which a face is pointing is defined by a vector called a *normal.* The direction of the normal indicates the front, or outer surface of the face.

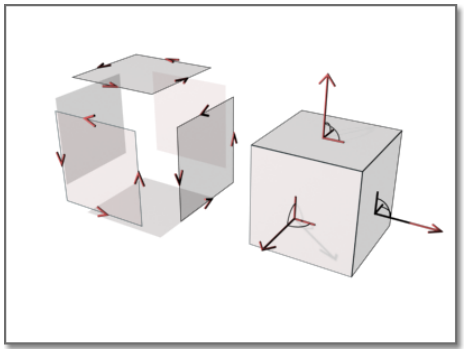


When normals are unified and point in the same outward direction, the renderer processes each face and renders the model. If any normals are flipped, facing inward, the renderer skips them and leaves triangular or quadrilateral "holes" in the rendered image.

In instances where you see a hole, it usually means one of two things: Force 2-Sided is turned off in the Render Settings palette or the face is physically missing from the model.

If a face is missing, you'll need to manually reconstruct it. The direction of normals is determined by the way a face is drawn in a right-handed coordinate system: if you draw the face counter-clockwise, the normals point outward; if you draw the face clockwise, the normals point inward. You should draw faces consistently.

NOTE Solid objects have meshes and normals correctly oriented, which can be an aid to creating models for rendering.



When rendering, the renderer searches for all normals that point away from the viewpoint and removes the associated faces from the scene. This removal step is called *back-face culling* and is controlled by the Force 2-Sided option on the Render Settings palette.

After the back faces have been removed, the renderer uses a *Z* buffer to compare relative distances along the *Z* axis. If the *Z* buffer indicates that one face overlaps another, the renderer removes the face that would be hidden.

The time saved is in proportion to the number of faces discarded out of the total number of faces.

Sometimes you may want to skip the back-face culling step and leave back faces in (for example, if an object is transparent, if you can see two sides of it because of its shape and orientation, or if an open object will be rendered with a view angle that lets you see inside). Transparency also affects whether one face should hide another. In this situation, make sure Force 2-Sided is active and all faces are rendered no matter which direction their normal is pointing.

If you're rendering a drawing that wasn't created with rendering in mind or if the model was created with another product, you should keep Force 2-Sided active. This ensures that all surfaces render correctly.

NOTE The rendering performance is only marginally affected when Force 2-Sided is active.

Every object in a scene is processed by the renderer, even objects that are off camera and are not going to be present in the rendered view. A model that is built with the intent of rendering will benefit from good layer management.

By turning off layers containing objects that are not in the view, you can increase rendering speed substantially.

See also:

- Use Models with Other Applications on page 2283
- Use a Visual Style to Display Your Model on page 294

To make sure both normals of a face get rendered

1. Click Render tab ➤ Render panel ➤  ➤ Advanced Render Settings.



2. In the Materials section of the Advanced Render Settings palette, make sure Force 2-Sided in set to On.
3. Render the scene.

Quick Reference

Commands

RPREF

Displays or hides the Advanced Render Settings palette for access to advanced rendering settings.

Minimize Intersecting and Coplanar Faces

Certain kinds of geometry create special rendering problems.

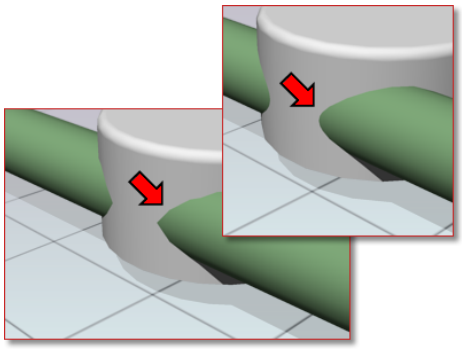
The complexity of an object relates to the number of its vertices and faces. The more faces a model has, the longer it takes to render. Keep the geometry of your drawing simple to keep rendering time to a minimum. Use the fewest faces possible to describe a surface.

Intersecting Faces

Intersecting faces in a model occur when two objects pass through one other. For conceptual design situations, simply placing one object through another

is a fast way to visualize how something will look. However, the edge created where the two objects intersect can exhibit a rippled appearance.

In the following example, the edge appears rippled in the left image and much cleaner after a Boolean union.

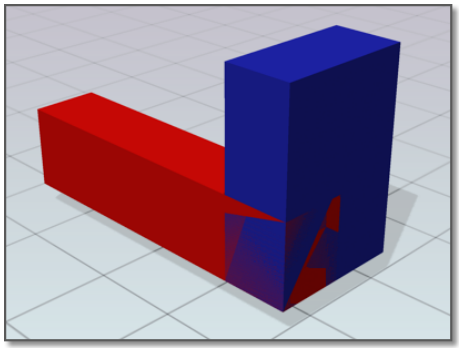


When edges do not appear to be as precise as you want, use Boolean operations like union, intersect, and subtract. A much cleaner and precise edge is created to better reflect the object's appearance.

Coplanar Faces

Faces that overlap and lie in the same plane, *coplanar faces*, can produce ambiguous results, especially if the materials applied to the two faces differ.

In the following example, artifacts appear when faces occupy the same location.

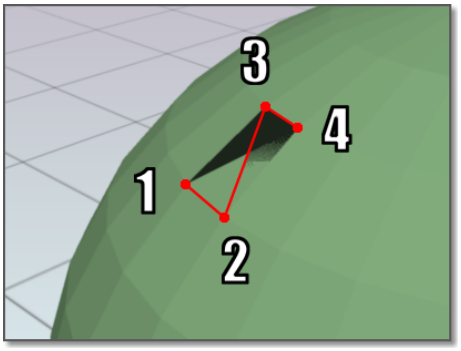


Moving an object so its faces no longer occupy the same plane as another object will fix this situation.

Twisted Faces

Faces that self-overlap due to a 180-degree twist can also produce ambiguous results, because the normal for the face is not well defined.

In the following example, artifacts appear where the face is twisted due to crossing the second and third corner points.



This situation is often encountered when trying to fix a model that has a hole in its surface. For example, when corner points are selected for the new face, the points are crossed instead of being placed around the hole in a counter-clockwise direction. Avoid this problem by choosing corner points in the proper order.

Quick Reference

Commands

RENDER

Creates a photorealistic or realistically shaded image of a 3D solid or surface model.

Balance Mesh Density for Smooth Geometry

When you render a model, the density of the mesh affects the smoothness of surfaces.

Mesh components are comprised of vertices, faces, polygons, and edges.

- A vertex is a point that forms the corner of a face or polygon.
- A face is a triangular portion of a surface object.

- A polygon is a quadrilateral portion of a surface object.
- An edge is the boundary of a face or polygon.

In a drawing, all faces have three vertices, except faces in polyface meshes, which are treated as adjoining triangles. For rendering purposes, each quadrilateral face is a pair of triangular faces that share one edge.

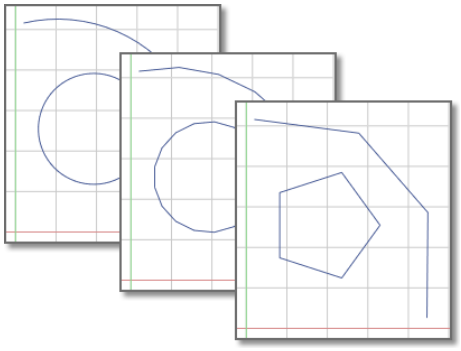
Smoothing of an object is handled automatically by the renderer. Two types of smoothing occur during the rendering process. One smoothing operation interpolates the face normals across a surface. The other operation takes into account the number of faces, the *face count*, that make up the geometry; greater face counts result in smoother surfaces but longer processing times.

While you cannot control the interpolation of face normals, you can control the display accuracy of curved objects by using the VIEWRES command and the FACETRES system variable.

Control Display of Circles and Arcs

The VIEWRES command controls the display accuracy of curved 2D linework like circles and arcs in the current viewport.

In the following example, line segments are more apparent as VIEWRES decreases - Upper left = 1000, Middle = 100, Lower right = 10.

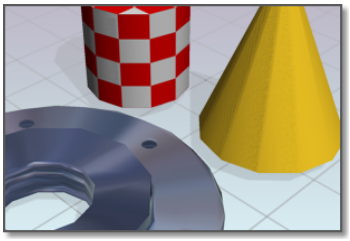


These objects are drawn on the screen using many short straight line segments. Smoother arcs and circles display with higher VIEWRES settings, but they take longer to regenerate. To increase performance while you're drawing, set a low VIEWRES value.

Control Display of Curved Solids

FACETRES controls the mesh density and smoothness of shaded and rendered curved solids.

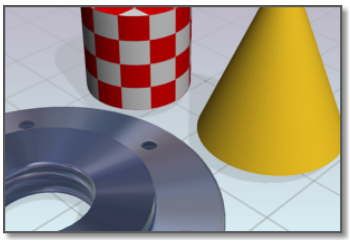
In the following example, facets display on curved geometry when FACETRES is low. FACETRES = .25.



When FACETRES is set to 1, there is a one-to-one correlation between the viewing resolution of circles and arcs and the tessellation, a means of subdividing the faces of solid objects. For example, when FACETRES is set to 2, the tessellation will be twice the tessellation set by VIEWRES. The default value of FACETRES is 0.5. The range of possible values is 0.01 to 10.

When you raise and lower the value of VIEWRES, objects controlled by both VIEWRES and FACETRES are affected. When you raise and lower the value of FACETRES, only solid objects are affected.

In the following example, smoother geometry is displayed when FACETRES is set to higher values. FACETRES = 5.



See also:

- Create Meshes on page 1111

To alter the render resolution of solid geometry

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

2 Do one of the following:

- Enter a value greater than .5 to increase the smoothness of curved surfaces.
- Enter a value lower than .5 to decrease the smoothness of curved surfaces.

To alter the display resolution of arcs and circles

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

2 Ignore the prompt about fast zooms if you only want to make circles and arcs in the drawing look better for your rendering.

3 At the Circle Zoom Percent prompt, do one of the following:

- Enter a value greater than 1000 to increase the smoothness of arcs and circles.
- Enter a value lower than 1000 to decrease the smoothness of arcs and circles.

Quick Reference

Commands

VIEWRES

Sets the resolution for objects in the current viewport.

System Variables

FACETRES

Adjusts the smoothness of shaded and rendered objects and objects with hidden lines removed.

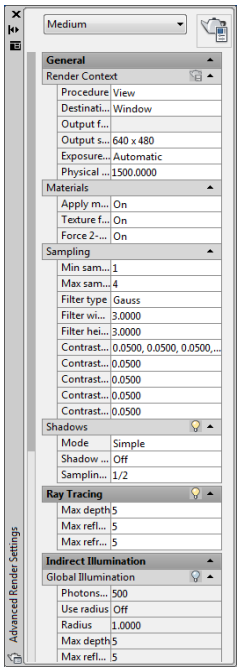
Set Up the Renderer

You can control many of the settings that affect how the renderer processes a rendering task, especially when rendering higher quality images.

Use the Render Settings Palette

The Render Setting palette contains the main controls for the renderer. You can choose from predefined render settings or make custom settings.

The RPREF command opens the Render Settings palette, where you set the parameters for rendering.



The palette is separated into several sections ranging from basic to advanced settings. The General sections contain settings that affect how your model gets rendered, how materials and shadows are handled, and how anti-aliasing is performed. (Anti-aliasing smooths the stairstep effect at the edge of curved lines or edges.) The Raytracing section controls how shading occurs. The Indirect Illumination section controls lighting properties, how your scene is illuminated, and if global illumination and final gathering are processed. There are also diagnostic controls that can be useful in helping you understand why an image isn't getting rendered as expected.

Render Presets

A drop-down list provides a collection of predefined render settings called *render presets*. Render presets store groups of settings that allow the renderer

to produce varying quality images. The standard presets range from Draft quality, for quick test images, up to Presentation quality, which provides photorealistic images. You can also open the Render Presets Manager where you can create custom presets.

See also:

- Create Custom Render Presets on page 2251

To create a custom render preset from the Advanced Render Settings palette

1. At the Command prompt, enter **rpref**.
2. In the Advanced Render Settings palette, select an existing standard or custom preset.
3. Make changes to the settings that you'd like to use for your rendering.

   Notice the original render preset has an '*' prefix indicating that it has been changed.
4. Render the model.
5. To save the render settings you made as a new, custom render preset, do one of the following:
   - Click in the preset name field at the top of the Advanced Render Settings palette and enter a new name.
   - Click in the render preset field on the Render panel of the ribbon and enter a new name.

Quick Reference

Commands

RPREF

Displays or hides the Advanced Render Settings palette for access to advanced rendering settings.

RPREFCLOSE

Closes the Render Settings palette.

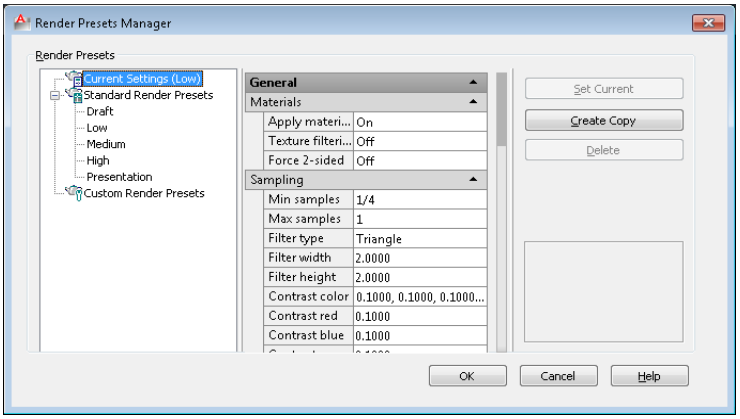
RENDERPRESETS

Specifies render presets, reusable rendering parameters, for rendering an image.

Create Custom Render Presets

When you specify a collection of render settings that give you the results you want, saving them as a custom preset lets you quickly reuse the settings.

Using a standard preset as a base, you can experiment with settings and see how the rendered images look. Once you're satisfied with the results, you can create a new, custom preset.



You can review and change the render settings for any preset in your drawing in the following ways:

- Manage and organize existing render presets.
- Change parameters of standard or existing custom presets.
- Create, update, or delete custom render presets.
- Set a render preset to be used by the renderer.

The tree view pane shows all the presets that are stored with the current drawing. The tree view is split between the standard render presets and any custom presets you've created. You can change the order of standard and custom presets by dragging one above the other in the tree view. When you create a new preset, it is always added to the Custom Render Presets branch.

It will also appear in the Render Presets lists in the Render panel on the ribbon and the Render Settings palette.

Many of the render settings found on the Render Settings palette can also be set in the Render Presets Manager. Using a standard preset as a base, you can make adjustments to the settings and see how the rendered images look. Once you're satisfied with the results, you can create a new, custom preset.

Only custom render presets can be deleted. If you've selected a standard preset, the Delete button is inactive.

In order for a render preset to be used by the renderer, it must be made current. While working in the Render Presets Manager, you select presets you want to edit, but in order for the renderer to use them, you have to click the Set Current button. Otherwise, the preset chosen on the Render panel on the ribbon or the Advanced Render Settings palette is used.

If you make changes to a standard render preset, its name is prefaced with an asterisk to indicate that a change has been made to its original settings. If the rendered results are acceptable, you can save the preset by entering a new name in the presets list on the Render panel on the ribbon or on the Render Settings palette.

If you change the settings of a custom render preset, you can enter a new name to create a new, custom render preset or choose to update the preset from the presets list on the Render panel on the ribbon or on the Render Settings palette.

To create a custom render preset from the Render Presets Manager

1. Click Render tab ➤ Render panel ➤ Render Presets drop-down ➤ Render Presets Manager.
2. In the Render Presets Manager, select an existing standard or custom preset.
3. Click the Create Copy button and specify a name for your new preset. Click OK.

   The new preset appears in the tree view under the Custom Render Presets branch.

To edit a custom render preset

1. Click Render tab ➤ Render panel ➤ Render Presets drop-down ➤ Render Presets Manager.
2. In the Render Presets Manager, select an existing custom preset.

3 Make adjustments from the render settings section.

   The preset name is prefaced with an asterisk to show it has been changed.

4 Click the Set Current button and render the scene.

   If the results are good, you can choose to update the current preset or re-open the Render Preset Manager and create a new, custom preset.

Quick Reference

Commands

RPREF

Displays or hides the Advanced Render Settings palette for access to advanced rendering settings.

RENDERPRESETS

Specifies render presets, reusable rendering parameters, for rendering an image.

Control the Rendering Environment

You can use environmental features to set up atmospheric effects or background images.

You can enhance a rendered image by means of atmospheric effects like fog and depth cueing or by adding a bitmap image as a background.

Fog / Depth Cue Effects

Fog and depth cueing are very similar atmospheric effects that cause objects to appear to fade as they increase in distance from the camera. Fog uses a white color while depth cueing uses black.



The RENDERENVIRONMENT command is used to set up fog or depth cue parameters. The key parameters you'll set are the color of the fog or depth cueing, the near and far distances, and the near and far fog percentages.

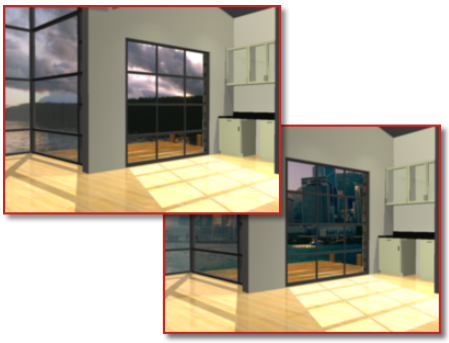
Fog and depth cueing are based on the front or back clipping planes of your camera coupled with the near and far distance settings on the Render Environment dialog box. For example, the back clipping plane of a camera is active and located 30 feet from the camera location. If you want fog to start 15 feet from the camera and spread away indefinitely, you set the Near Distance to 50 and the Far Distance to 100.

The density of the fog or depth cueing is controlled by the Near and Far Fog Percentages. These settings have a range of 0.0001 to 100. Higher values mean the fog or depth cueing is more opaque.

TIP For smaller scale models, the Near and Far Fog Percentage setting may need to be set below 1.0 to see the desired effect.

Backgrounds

A background is basically a backdrop that displays behind your model. Backgrounds can be a single color, a multi-color gradient, or a bitmap image.



Backgrounds work best when you are rendering still images, or animations in which the view doesn't change or the camera doesn't move. Once set, the background is associated with the named view or camera and is saved with the drawing.

You set backgrounds from the View Manager.

To use fog / depth cueing

1. Select a camera and open its Property palette.
2. Turn on its front or back clipping plane.
3. Set the clipping plane offset value.
4. Click Visualize tab ➤ Render panel ➤ Environment. 
5. Turn on Enable Fog and set the color of the fog or depth cueing.

   Light colors are typically used for fog while dark colors are used for depth cueing.
6. Set the Near and Far Distance values to specify where the fog starts and ends.
7. Set the fog or depth cue opacity with the Near and Far Fog Percentage values. Click OK.
8. Render the model.

To set a background for a named view

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

2. In the View Manager, select an existing named view from the Model Views list.
3. On the General panel, click the Background Override list and select Solid, Gradient, or Image.
4. In the Background dialog box, set the colors or choose a bitmap image to use for the background. Click OK.
5. Click OK to close the View Manager.
6. On the ribbon, select the named view from the drop-down list on the View tab ➤ Viewports panel.
7. Render the scene.

To edit a background for a named view

1. At the Command prompt, enter **view**.
2. In the View Manager, select an existing named view from the Model Views list.
3. On the General panel, click the Background Override list and select Edit.
4. In the Background dialog box, set the new colors or choose a new bitmap image to use for the background. Click OK.
5. Click OK to close the View Manager.
6. On the ribbon, select the named view from the drop-down list on the View tab ➤ Viewports panel
7. Render the scene.

Quick Reference

Commands

RENDERENVIRONMENT

Controls visual cues for the apparent distance of objects.

VIEW

Saves and restores named model space views, layout views, and preset views.

Basics of Rendering

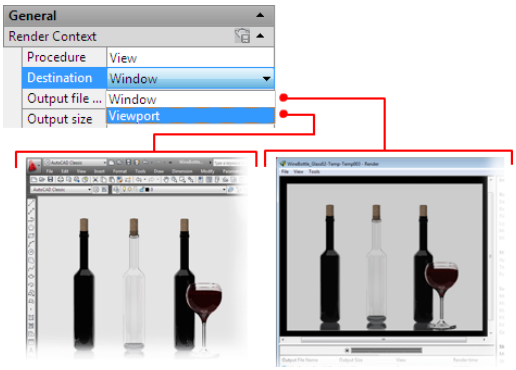
While the final goal is to create a photorealistic, presentation-quality image that illustrates your vision, you create many renderings before you reach that goal.

At a basic level, you can use the RENDER command to render your model without applying any materials, adding any lights, or setting up a scene. When you render a new model, the renderer automatically uses a virtual "over-the-shoulder" distant light. You cannot move or adjust this light.

Set the Render Destination

Rendered images appear either in a viewport or get displayed in the Render Window.

When you render a scene, the image can be displayed in either the viewport or the Render Window. This is the *render destination*.



The render destination is set in the Advanced Render Settings palette in the Render Context section. The default setting is Window.

When the render destination is set to Window, the renderer automatically opens the Render Window and the image is processed. Upon completion, the image is displayed and a history entry is created. As more renderings occur, they are added to the render history so you can quickly look at previous images and compare to see which have the desired results. Images that you want to keep can be saved from the Render Window.

If you choose to set the render destination to Viewport, the generated image is rendered and displayed directly in the active viewport. In essence, this is a one-time rendering because there is no render history entry that you can

compare with later images. If you want to keep the image that you rendered to the viewport, you can use the SAVEIMG command to save the images.

Rendering to a viewport always renders against the background color you set for the drawing area. The Render Window background color matches the background color. Use the REGEN command to refresh the display.

See also:

- Render Window

To set the render destination

1. At the Command prompt, enter **rpref**.
2. In the Advanced Render Settings palette, scroll to the top of the palette. Under General, open the Render Context rollout, if necessary.
3. Open the Destination list and select Window or Viewport.
4. Render the scene.

Quick Reference

Commands

RENDER

Creates a photorealistic or realistically shaded image of a 3D solid or surface model.

RPREF

Displays or hides the Advanced Render Settings palette for access to advanced rendering settings.

SAVEIMG

Saves a rendered image to a file.

Render Views, Selected Objects, or Cropped Content

You can render an entire view, a set of selected objects, or a portion of what you see in the viewport.

Render a View

The default rendering procedure is to render all objects in the current view in the drawing. If you haven't opened a named view or a camera view, the current view is rendered. While the rendering process is faster when you render key objects or smaller portions of a view, rendering the entire view lets you see how all objects are oriented to one another.

If your current drawing contains named views or if you've added cameras to your model, you can quickly display them by using the VIEW command.

The following example shows a rendering of a named view.



Depending on the rendering destination you've chosen, the rendered view is displayed in the Render Window or directly in the viewport. For a complete description of the Render Window, see RENDER in the *Command Reference*.

Render Selected Objects

If you're adding detail to specific objects, you don't want to waste time rendering an entire viewport. By changing the rendering procedure to Selected, you are prompted to pick the objects that you want rendered.

The following example shows a rendered selection of the first bottle, its cork, and the wine glass.

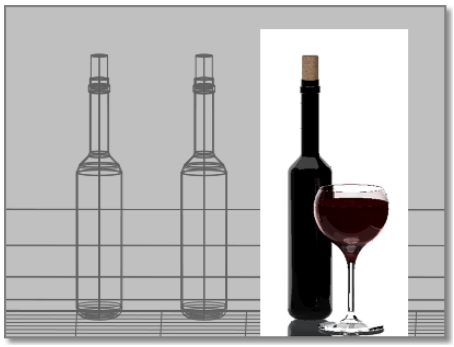


Rendering a selection set of objects is very efficient when testing different materials, especially when the materials include texture mapping. By rendering a selected object, you can quickly verify how the material looks and if its texture coordinates must be altered.

Render a Cropped Region

Sometimes you need to render only a portion of what is displayed in the viewport but you still want to see some of the surrounding environment.

In the following example, only the region surrounding the third bottle and the wine glass is rendered.



By setting the rendering procedure to Crop, you can specify a smaller region of the viewport to be rendered. Similar to selecting objects by window, you can set a rectangular region in the viewport. Any objects that appear in the region are rendered. Everything outside the region is ignored by the renderer.

NOTE A cropped rendering only displays in the viewport. Use the REGEN command to refresh the display.

To render a view

1. Display a 3D view of your model.
2. Click Render tab ➤ Render panel ➤  ➤ Advanced Render Settings.

   
3. Choose a render preset to control the quality and speed of the rendered output.

   The Draft preset is very fast but its quality is low. Presentation generates a high quality image but takes substantially longer to render.
4. Set the Destination to Window or Viewport to specify where you want the rendered image to be displayed.

   If you set an image target, the image is saved directly to a file as well as appearing on screen.
5. Render the scene.

To render a selection set of objects

1. Click Render tab ➤ Render panel ➤  ➤ Advanced Render Settings.

   
2. Choose a render preset to control the quality and speed of the rendered output.

   The Draft preset is very fast but its quality is low. Presentation generates a high quality image but takes substantially longer to render.
3. Set the render procedure to Selected.
4. Select the objects in the model that you want to render.
5. Render the scene.

To render a cropped view

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

2 Specify a window in the viewport that you want to render.

3 Render the scene.

Quick Reference

Commands

RENDER

Creates a photorealistic or realistically shaded image of a 3D solid or surface model.

RENDERCROP

Renders a specified rectangular area, called a crop window, within a viewport.

RPREF

Displays or hides the Advanced Render Settings palette for access to advanced rendering settings.

Set Output Resolution

Set the resolution of the rendered image by specifying the width and the height of the image, in pixels.

There are three resolution settings that control how a rendered image appears; the width, the height, and the image aspect ratio.

The width and height settings control the size of the rendered image, measured in *pixels*. A pixel (short for Picture Element) is a single point in a graphic image.

The default output resolution is 640 x 480 and can be set as high as 4,096 x 4,096. Higher resolution settings result in smaller pixels and finer detail. High resolution images also take longer to render.

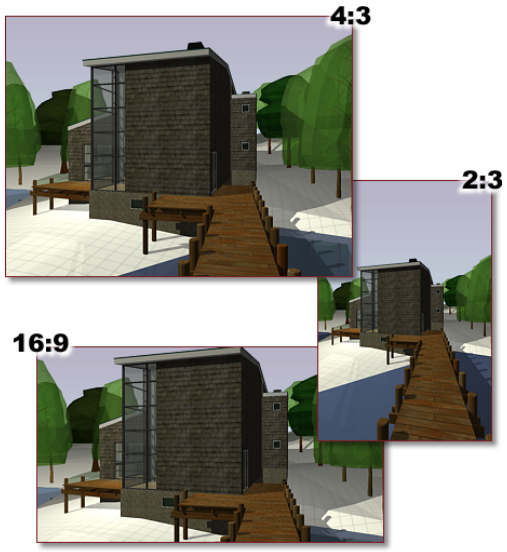
Output resolutions are set from the Output Size Dialog Box. You can enter values directly into the width and height fields, or you can use the spinner controls to increase or decrease the resolution.

When you set an output resolution, it gets stored with the current drawing and is added to the output resolution list found in the Render panel of the ribbon. Most often, as you test how objects look in the model, you will find yourself using lower resolution settings, around 320 x 200 or lower. As you add more detail and materials, you'll shift to mid-range settings, such as 640 x 480. The final rendering will always use the highest resolution required by

the project, 1024 x 768 or greater, since this is the image that is presented to the customer or submitted for print.

Aspect ratio describes the proportions of a still image or the frames in an animation, expressed as the ratio of width to height, regardless of the image's resolution. The aspect ratio of an image is controlled by the Image Aspect setting. Aspect ratio is usually expressed either as a ratio of width over height (for example, 4:3) or as a multiplier (such as, 1.333). Changing this value changes the Height value to maintain the correct dimensions for the output resolution.

The following example shows various aspect ratios.



If you choose to lock the image aspect, the width and height are tied together; changing one automatically changes the other while maintaining the aspect ratio.

Quick Reference

Commands

RENDER

Creates a photorealistic or realistically shaded image of a 3D solid or surface model.

RPREF

Displays or hides the Advanced Render Settings palette for access to advanced rendering settings.

Material Adjustments

Adding materials to objects greatly increases the realism of a model.

In the context of rendering, materials describe how an object reflects or transmits light. Within a material, maps can simulate textures, bump effects, reflections, or refractions.

From the Advanced Render Settings palette, you can turn materials on or off, turn material filtering on or off, and affect how the surfaces of an object are rendered.

Materials that you've created and attached to objects in the model are normally turned on when you start the rendering process. If you turn them off, all the objects in the model assume the characteristics of the GLOBAL material.

Texture Filtering

You can control whether texture filtering, pixel averaging used in anti-aliasing, occurs when the model is rendered.

When texture filtering is turned on, the renderer uses a pyramidal filtering methods, which applies filtering as a function of distance.

NOTE Irregular anti-aliasing might occur on detailed texture maps that are applied to a surface that recedes into the distance.

There is a slight rendering cost when texture filtering is active. Pyramidal filtering requires memory allocation equal to approximately 133% of the size of a texture map. Compared to the detail and realism provided by texture mapped materials, rendering performance is not impacted that much.

When texture filtering is turned off, no anti-aliasing occurs to texture maps when the renderer processes the model.

2-Sided Rendering

Since you have no control over the direction that faces, normals are pointing, you may encounter a model that has flipped normals. The back sides of faces are invisible to the renderer. This means that the face appears to be missing when viewed from the back. Objects are usually created with the surface normals facing outward, but it is possible to create objects with the faces

flipped or to import complex geometry in which the face normals are not properly unified.

When Force 2-Sided is active, both sides of faces are raytraced as shaded. While this incurs a slight increase in rendering time, it is often faster than trying to fix multiple instances of flipped faces.

Quick Reference

Commands

MATERIALATTACH

Associates materials with layers.

MATERIALMAP

Displays a material mapping gizmo to adjust the mapping on a face or an object.

MATERIALS

Opens the Materials Browser.

RENDER

Creates a photorealistic or realistically shaded image of a 3D solid or surface model.

RPREF

Displays or hides the Advanced Render Settings palette for access to advanced rendering settings.

System Variables

LOGEXPPHYSICALSCALE

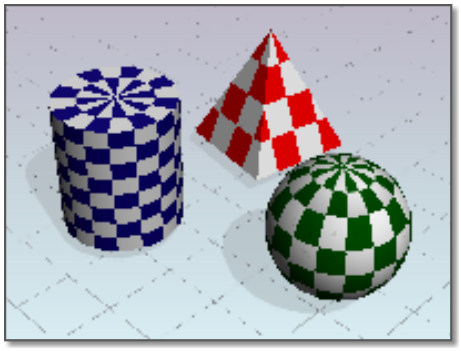
Controls the relative brightness of self-illuminated materials in a photometric environment.

Adjust Sampling to Improve Image Quality

You can use several standard render presets to remove the jagged effect from the display of diagonal lines and curved edges.

Because the image on a monitor is made up of discrete pixels on a fixed grid, scenes rendered using the Draft or Low render presets produce jagged and inaccurate images.

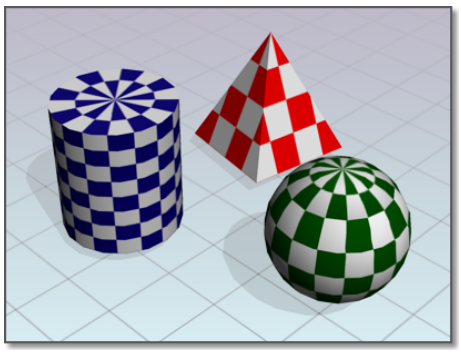
In the following example, image quality is highly degraded - Min. Samples = 1/64; Max. Samples = 1/4.



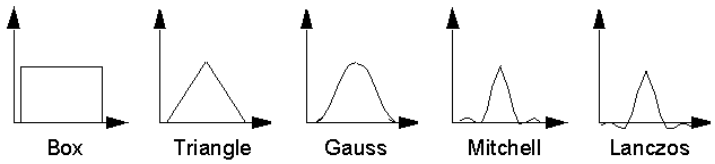
The greater the resolution (and thus the smaller the pixels), the less apparent aliasing is. However, it is often best to reduce the effect further by using *anti-aliasing* techniques. The anti-aliasing technique used by the renderer is called *sampling*. Sampling is controlled by the sample and filter settings on the Advanced Render Settings palette.

Sampling 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. Increasing the minimum and maximum sample ranges greatly improves the quality of a rendering.

In the following example, increased sampling smooths edges - Min. Samples = 1; Max. Samples = 16.



There are five filter methods: Box, Gauss, Triangle, Mitchell, or Lanczos. Box, the default, is the quickest. Mitchell is often the most accurate. The Box filter combines samples evenly, without weighting them. Each of the other filters uses a particular curve to weight samples before combining them.



Anti-aliasing techniques involve at least two trade-offs.

- **Extra calculation**. Pixels in the offending line or boundary and neighboring pixels in the background must be analyzed so that intermediate shades can be added.
- **Thicker lines**. Although an anti-aliased line appears smoother, it must be thicker than a jagged one.

You have to decide how much time you want to spend rendering and how good you want the final rendering to be. These trade-offs will depend partly on the hardware you're using and partly on the audience for your work.

The standard render presets offer five levels of anti-aliasing. Draft is set to use the lowest level of anti-aliasing, but it provides the fastest rendering speed. Each successive level is slower to compute. The anti-aliasing algorithm makes the process more efficient by allowing the program to decide for any given pixel that fewer samples need to be computed.

Quick Reference

Commands

RENDER

Creates a photorealistic or realistically shaded image of a 3D solid or surface model.

RPREF

Displays or hides the Advanced Render Settings palette for access to advanced rendering settings.

Use Shadows in Rendering

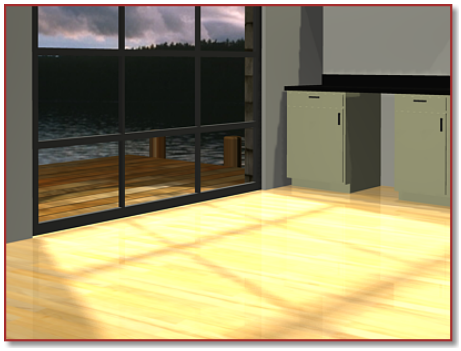
With shadows, you can create rendered images that have greater depth and realism.

The renderer can generate shadows by shadow mapping or by ray tracing. Shadow mapped shadows rely on a bitmap that the renderer generates during a pre-rendering pass of the scene. Shadow mapping provide softer edges and can require less calculation time than ray-traced shadows, but are less accurate. 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 more accurate, hard edges and require more calculation time.

Shadow-Mapped Shadows

Shadow maps are the only way to generate soft-edged shadows, but they do not show the color cast by transparent or translucent objects. Shadow-mapped shadows are calculated faster than ray-traced shadows.

The following example shows how shadow maps produce softer, less precise shadows that are processed faster than raytraced shadows.

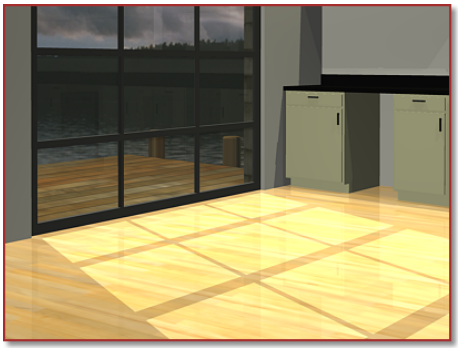


During a pre-rendering pass, a shadow map bitmap is created. Shadow quality can be controlled by increasing or decreasing the size of the shadow map. The default shadow map size is 256 x 256 pixels. If the shadow appears to be too grainy, increasing the map size will give you better quality. Shadow mapped shadows should not be used if you have a light shining through a transparent surface (for example, a multi-pane window where you want the the frame and mullions to cast shadows). You'd have to remove the glass in order for the mullions to cast shadows.

Ray-traced Shadows

Ray-traced shadows (like other ray-traced effects of reflection and refraction) are generated by tracing the path of light beams or rays sampled from a light source. Ray-traced shadows are more accurate than shadow-mapped shadows.

The following example shows that while ray tracing takes longer to process, it produces more realistic, precise shadows.



Ray-traced shadows have hard edges and accurate outlines; they also transmit color from transparent and translucent objects. Therefore, shadows of the frame and mullions of a multi-pane window will render. Because ray-traced shadows are calculated without a map, you don't have to adjust resolution as you do for shadow-mapped shadows.

Shadow Modes

You can select one of three shadow mode settings when shadows are turned on. The shadow mode can be set to Simple, Sort, or Segment.

- **Simple**. The renderer calls shadow shaders in a random order. This is the default mode state for shadows.
- **Sorted**. The renderer calls shadow shaders in order, from the object to the light.
- **Segment**. The renderer calls shadow shaders in order along the light ray from the volume shaders to the segments of the light ray between the object and the light.

Displaying Shadows

In order for shadows to be cast in a model, lighting must be established. A light source needs to be added to the scene and you need to specify if that

light source will cast shadows. For shadows to display in the viewport as you set up the scene, you need to turn on shadows for the visual style. If you want shadows to appear in the rendered image, you need to turn on shadows and choose the type of shadows to render on the Advanced Render Settings palette.

To view shadows in the viewport

1. Click View tab ➤ 3D Palettes panel ➤ Visual Styles. 
2. In the Visual Style Manager, select either Conceptual or Realistic. Shadow display has no effect for 3D Hidden or 3D Wireframe styles.
3. Set Shadow Display to Ground Shadows or Full Shadows.
4. Close the Visual Style Manager.

To generate shadow mapped shadows in the rendered image

1. Click Render tab ➤ Render panel ➤  ➤ Advanced Render Settings. 
2. In the Advanced Render Settings palette, make sure Shadows is turned on.
3. Select the shadow mode you want to use.
4. Turn on Shadow Map.
5. Render the model.

To generate ray-traced shadows in the rendered image

1. Click Render tab ➤ Render panel ➤  ➤ Advanced Render Settings. 
2. In the Advanced Render Settings palette, make sure Shadows is turned on.
3. Select the shadow mode you want to use.

4 Turn off Shadow Map.

5 Render the model.

Quick Reference

Commands

RENDER

Creates a photorealistic or realistically shaded image of a 3D solid or surface model.

RPREF

Displays or hides the Advanced Render Settings palette for access to advanced rendering settings.

VISUALSTYLES

Creates and modifies visual styles and applies a visual style to a viewport.

Advanced Rendering

Advanced rendering techniques allow you to render highly detailed and photorealistic images.

Ray-traced Reflections and Refractions

Ray tracing traces the path of rays sampled from the light source. Reflections and refractions generated this way are physically accurate.

To reduce the time required to generate reflections and refractions, rays are limited by *trace depth*. Trace depth limits the number of times a ray can be reflected, refracted, or both. You set the maximum trace depth, the maximum number of reflections, and the maximum number of refractions in the Advanced Render Settings palette.

The following example shows how ray-traced reflections and refractions greatly increase the realism of a scene. Max Depth = 8; Max Reflections and Refractions = 4.



When ray tracing is turned off, no reflection or refraction occurs.

The following example shows the same model with ray tracing turned off.



Trace depth controls the number of times a light ray can be reflected or refracted. Increasing these values can increase the complexity and realism of a rendered image, at a cost of greater rendering time.

The following example shows how increasing the trace depth improves the rendering. Max Depth = 4; Max Reflections and Refractions = 2.



Ray-tracing Controls

The Max Depth setting limits the combination of reflection and refraction. Tracing of a ray stops when the total number of reflections and refractions reaches the maximum depth. For example, if Max Depth equals 3 and Max Reflections and Max Refractions are both set to 2, a ray can be reflected twice and refracted once, or vice versa, but the ray cannot be reflected and refracted four times.

The Max Reflections setting specifies the number of times a ray can be reflected. At 0, no reflection occurs. At 1, the ray can be reflected once only. At 2, the ray can be reflected twice, and so on.

The Max Refractions setting specifies the number of times a ray can be refracted. At 0, no refraction occurs. At 1, the ray can be refracted once only. At 2, the ray can be refracted twice, and so on.

NOTE As you increase the number of reflections and refractions, you will also increase the amount of processing time required by the renderer.

Quick Reference

Commands

RENDER

Creates a photorealistic or realistically shaded image of a 3D solid or surface model.

RPREF

Displays or hides the Advanced Render Settings palette for access to advanced rendering settings.

Benefits of Indirect Illumination

Indirect illumination techniques, like global illumination and final gathering, enhance the realism of a scene by simulating radiosity, or the interreflection of light in a scene.

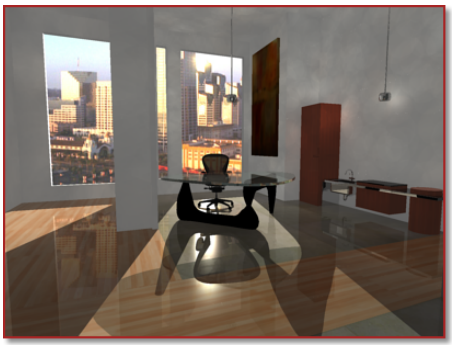


Global illumination (GI) provides effects such as color bleeding. For example, if a red countertop is next to a white wall, the white wall gets a slightly pink tint. You might also think it's a minor detail, but if the pink tint is missing from the image, the image looks fake, even though it might be hard to point out precisely why. This effect is not possible with ordinary ray-tracing calculations.

To calculate global illumination, the renderer uses *photon maps*, a technique to generate the indirect illumination effects and global illumination. A side effect of using a photon map is 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.

When preparing for a finished rendering, make sure you have specified the drawing units you want to use before you make any GI settings. Changing drawing units after you're satisfied with the global illumination will adversely affect the rendering results.

The accuracy and intensity of global illumination is controlled by the number of photons generated, the sampling radius, and its trace depth. This example shows the effects of a low number of photons and a sampling radius set too small.



Photons and Sampling Radius

The intensity of global illumination is computed by the number of photons you've specified. Increased numbers of photons makes global illumination less noisy but also more blurry. When the number of photons are decreased, global illumination is more noisy but less blurry. The greater the number of photons, the longer the rendering time.

TIP To preview global illumination, give settings such as Photons/Sample or Photons/Light lower values, then increase the values for the final rendering.

The sampling radius sets the size of photons. In many cases, the default photon size (Use Radius=Off) of one-tenth the scene size gives useful results. In other cases, the default photon size might be too large or too small.

The size of the sampling radius (Radius) determines if photons overlap. When photons overlap, the renderer smooths them together. Increasing the radius increases the amount of smoothing and can create more natural-looking illumination. When photons have a small radius and don't overlap, no smoothing occurs. Ideally, photons should overlap. To get good results, you would want to turn on Use Radius and increase the Radius size.

Global Illumination Trace Depth

The Trace Depth controls are similar to those for calculating reflections and refractions, but they refer to the photons used by global illumination rather than to light rays used in ray-traced reflection and refraction.

The maximum depth limits the combination of reflection and refraction. Reflection and refraction of a photon stops when the total number of both equals the maximum depth. For example, if Max Depth equals 3 and Max Reflections and Max Refractions are both set to 2, a photon can be reflected

twice and refracted once, or vice versa, but it can't be reflected and refracted four times.

The Max Reflections setting specifies the number of times a photon can be reflected. At 0, no reflection occurs. At 1, the photon can be reflected once only. At 2, the photon can be reflected twice, and so on.

The Max Refractions setting specifies the number of times a photon can be refracted. At 0, no refraction occurs. At 1, the photon can be refracted once only. At 2, the photon can be refracted twice, and so on.

Quick Reference

Commands

RPREF

Displays or hides the Advanced Render Settings palette for access to advanced rendering settings.

Final Gathering

Final gathering is an optional, additional step used to improve global illumination (GI). It increases the number of rays used to calculate GI to smooth out and eliminate adverse lighting artifacts.

Because global illumination is calculated from a photon map, rendering artifacts such as dark corners and low-frequency variations in the lighting may occur.



By activating final gathering, you increase the number of rays used to calculate global illumination, which reduces or eliminates these artifacts.



There is a catch when deciding to use final gathering. Final gathering can greatly increase rendering time. It is most useful for scenes with overall diffuse lighting.

TIP Leave final gathering off to preview the scene, then turn it on for the finished rendering. (Increasing the number of photons used to calculate global illumination can also improve global illumination.)

You activate and adjust final gathering in the Advanced Render Settings palette.

Quick Reference

Commands

RPREF

Displays or hides the Advanced Render Settings palette for access to advanced rendering settings.

Rendered Image Histories

The render history maintains a list of recently rendered images generated from the currently loaded model.

Each image that is rendered to a file or the Render Window is recorded with the drawing as a *history entry*.

Understand History Entries

History entries store data pertaining to renderings made from the current drawing.

Each history entry contains information about the images that have been recently rendered. The information stored in history entries includes

- Date and time of the rendering task
- Statistics for the rendering task
- Current view or the name of a stored view
- Active render preset name
- Active render settings, such as output resolution
- Size of a cropped region, if crop was used for the render task

NOTE Rendered images are not stored in the history entry, only a link to the image location.

History Entry Information

Aside from the statistics, history entries contain five pertinent pieces of information.

Output File Name Holds two pieces of information: the name of the rendering and its state. If you specify an output file, that name is listed in the field. If you have not specified an output file, the drawing name is listed with a temporary suffix.

Output Size Tells you the output resolution for the rendered image.

View Is always the current view unless you've chosen to render a saved view. In that case, the view name is displayed.

Render Time Shows the date when the image was rendered and the amount of time it took to render.

Render Preset Displays whatever render preset was used by the renderer to generate the image.

History Entry Creation

There are some rules that delegate when a history entry is created and if that history entry is stored with the current drawing. The destination setting is the primary factor that determines if a history entry is created. The following table

outlines the circumstances of when a history entry is created and when it is saved.

Destination	History entry created?	Entry saved with drawing?
File	Yes	Yes
Render Window	Yes	Only if the Render Window view is saved
Viewport	No	Not applicable

Any history entries that are generated, but not saved with the drawing only exist for the duration of the current drawing session. Once you close the file or exit the program, unsaved history entries are lost.

History Entry States

There are four states in which a history entry can appear in the Render Window.

Normal When you specify an output location and file name for a rendering, the history entry is created and saved with the drawing. This history entry is in a normal state and the file name you specified is displayed in the File Name column of the render history section of the Render Window.

Temporary A temporary history entry occurs when you render an image only to the Render Window without specifying an output file. This history entry remains in memory until the drawing session ends. Temporary history entries have a *-TempXXX* suffix appended to their Output File Name listing.

Empty When you specify an output location and file name for a rendering, the history entry is created and saved with the drawing. This history entry is in a normal state and the file name you specified is displayed in the Output File Name column of the render history section of the Render Window.

Quick Reference

Commands

RENDER

Creates a photorealistic or realistically shaded image of a 3D solid or surface model.

RENDERWIN

Displays the Render window without starting a rendering operation.

RPREF

Displays or hides the Advanced Render Settings palette for access to advanced rendering settings.

Save and Redisplay Rendered Images

You can save a rendering and then redisplay it later. Redisplaying is much faster than rendering again.

Save a Rendered Image

You can save an image of a model rendered to a viewport or a render window, or you can render the image directly to a file.

Depending on the render settings and render preset you've chosen, rendering can be a time-consuming process. However, redisplaying a previously rendered image is instantaneous.

To save a rendered image, you can render directly to a file, you can render to the viewport and then save the image, or you can render to the Render Window and save or save a copy of the image. Once an image is saved, you can view your images at any time. Saved images can also be used as texture maps for materials that you've created.

Render Directly to a File

No matter how your display is configured, you can bypass the screen and redirect your rendering to a file. An advantage of not rendering to the screen is that you can render to higher resolutions than your current display configuration permits. You can then view that image on other systems with higher-resolution displays. You set the file location and file name by setting the Image Target in the Advanced Render Settings palette or on the Render panel of the ribbon. Rendered images are saved to one of several file formats, such as BMP, TGA, TIF, PCX, JPG, and PNG.

Save a Viewport Rendering

You can choose to render to a viewport. After rendering a model to a viewport, you use SAVEIMG to save the displayed image to one of the following file formats: BMP, TGA, TIF, PCX, JPG, or PNG. You can save the file to an

assortment of different grayscale or color depths that are offered by the file format you've selected.

Save a Render Window Rendering

If you've chosen the render destination to be the render window, you can save the image or save a copy of the image to one of the following file formats: BMP, TGA, TIF, PCX, JPG, or PNG. Depending upon the file format you've selected, you can choose to save grayscale or color depths ranging from 8 bits to 32 bits per pixel (bpp).

To render an image directly to a file

1. Click Render tab ➤ Render panel ➤  ➤ Advanced Render Settings.

2. In the Advanced Render Settings palette, turn on Save Rendering to File.
3. Click the [...] button next to the Output File Name field.
4. In the Save As dialog box, select a file type and the location where you want the image saved. Click Save.
5. Click Render.

   As the rendering is completed, the image appears in the Render Window. It is also saved to the drive location you chose with the name you specified.

To save an image that is rendered to the viewport

1. After the image is rendered to the viewport, at the Command prompt, enter **saveimg**.
2. In the Render Output File dialog box, browse to a folder where you want to store the image.
3. Enter a name for the image and an output file format.
4. Click Save.

To save a rendered image from the Render Window

1. Select a history entry from the Render Window.

2 Click File menu ➤ Save.

3 In the Render Output File dialog box, select a file format, and enter a storage location and file name. Click Save.

The image is saved in the selected file format.

Quick Reference

Commands

RENDER

Creates a photorealistic or realistically shaded image of a 3D solid or surface model.

RENDERWIN

Displays the Render window without starting a rendering operation.

RPREF

Displays or hides the Advanced Render Settings palette for access to advanced rendering settings.

SAVEIMG

Saves a rendered image to a file.

Redisplay a Rendered Image

Having saved your rendered image, you can redisplay that rendering at any time.

Having saved your rendered image, you can view that rendering at any time by opening it in the Windows Image Viewer.

If you don't want to use an external program, you can also view the image by inserting it into the drawing with the IMAGEATTACH command.

Quick Reference

Commands

IMAGEATTACH

Inserts a reference to an image file.

RENDER

Creates a photorealistic or realistically shaded image of a 3D solid or surface model.

Save Copies of Rendered Images

You can save copies of images that are part of a drawing's render history.

With the Save Copy option, located in the Render Window, you can save a rendered image to a new location even though an output path may already be set. This is useful if you want to re-render a history entry and compare the new image with the previous.

To save a rendered image from the Render Window

1. Select a history entry from the Render Window.
2. Click File menu ➤ Save Copy.
3. In the Render Output File dialog box, select a file format, and enter a storage location and file name. Click Save.

   The image is saved in the selected file format.

Quick Reference

Commands

RENDER

Creates a photorealistic or realistically shaded image of a 3D solid or surface model.

RENDERWIN

Displays the Render window without starting a rendering operation.

RPREF

Displays or hides the Advanced Render Settings palette for access to advanced rendering settings.

Use Models with Other Applications

You can use other Autodesk products to further enhance the quality of your models.

Autodesk® products continually improve their ability to share drawings and models, including 3ds Max®, Autodesk® VIZ, AutoCAD® Architecture, and AutoCAD® Mechanical.

3ds Max or Autodesk VIZ

With these products, you can make greater improvements on your models. You have the option of opening DWG or DXF files without converting or you can use the File Link Manager to create a live link with a drawing file. 3ds Max or Autodesk VIZ offers expanded animation, lighting, material, and rendering capabilities that add further polish to your presentation graphics.

AutoCAD Architecture

AutoCAD geometry can be opened in AutoCAD Architecture, no conversion is necessary. Once the model is open you can use architectural related features, such as architectural objects, schedules or integrated rendering, to streamline your architectural design and documentation process.

AutoCAD Mechanical

If you want to use AutoCAD geometry and rendering data with AutoCAD Mechanical, you can export and import files using the following methods:

- To export a drawing, click File menu ➤ Export.
- To import a file, click Insert menu ➤ Drawing Exchange Binary.

Quick Reference

Commands

3DSIN

Imports a 3D Studio (3DS) file.

AMECONVERT

Converts AME solid models to AutoCAD solid objects.

Update Legacy Drawings

When you open drawings created with earlier versions of AutoCAD, content is converted to be usable in rendering.

Lights, Materials, and Cameras

Lights, materials, and cameras have been improved but some older drawings may contain AutoShade information, such as lights, scene blocks, and camera blocks.

Lights and materials used in older drawings can be converted. AutoCAD 2008 introduced the 3DCONVERSIONMODE system variable. With the system variable set to the default value of 1, lighting and materials are automatically converted for older drawings.

For more information about converting drawings, see Converting Lights on page 2200 and Convert Materials on page 2219 topics.

Quick Reference

Commands

RENDER

Creates a photorealistic or realistically shaded image of a 3D solid or surface model.

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.

3Dconnexion Set of navigation tools used to reorient the current view of a model with a 3Dconnexion device.

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.

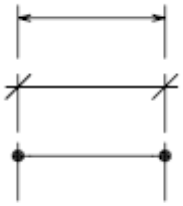
application menu The menu that is displayed when you click the application button. The application menu contains common tools for creating, saving, and publishing a file.

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)

associative surfaces Associative surfaces automatically adjust their location and shape when the geometric objects associated with them are modified. Controlled by the SURFACEASSOCIATIVITY system variable.

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.

bulge magnitude The amount of curvature where two surfaces meet. This only applies to surfaces that have G1 or G2 continuity.

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.cuix*).

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.

clamp curve A smooth, closed curve such as a circle. Because it has a vertex that is tangent to the object, if the curve is reshaped, it may create kinks. See also periodic curve.

clamp surface A smooth, closed surface such as a cylinder. Because as a vertex that is tangent to the object, if the surface is reshaped, it may create kinks. See also periodic surface.

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.

coincident grip Grip shared by multiple objects.

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* work plane.

contextual ribbon tab A ribbon tab that is displayed only when a particular type of object or when a particular command is executed. For example, selecting a hatch or table, or executing the mtext command brings up the corresponding contextual menu.

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)

continuity A measure of how smoothly two curves or surfaces flow into each other where they are joined. Continuity is measured as G0-Position, G1-Tangency, and G2-Curvature.

- **G0 (Position)** - The curves or surfaces join in the same location (position only); they touch. But the tangency and curvature do not match.
- **G1 (Tangent)** - The position and tangency between the surfaces match. This indicates G1 (position + tangency) continuity between the surfaces.
- **G2 (Curvature)** - The position, tangency, and curvature between the surfaces match. This indicates G2 (position + tangency + curvature) continuity between the two surfaces.

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 CVSHOW and CVHIDE commands must be turned on to display and hide control frames.

control point *See* control frame.

control vertices (CVs) The most basic way to shape a NURBS surface or spline. These points act as grips that can be dragged to reshape the object.

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.

curve-fit A smooth curve consisting of arcs joining each pair of vertices. The curve passes through all vertices of the polyline and uses any tangent direction you specify.

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 (CUIx) file An XML-based file that stores customization data. You modify a customization file through the Customize User Interface Editor. CUIx files replace MNU, MNS, MNC, and CUI files that were used to define menus in earlier releases.

CV hull A NURBS surface is modified through its control vertices (CV) hull. It consists of the control vertices and the lines that connect them in the U and V directions The hull sits outside of (not on) the surface. NURBS *curves* do not have a CV hull; they only have control vertices.

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.

degree A mathematical property of a curve or a surface that controls how many control vertices per span are available for modeling.

The higher the degree, the more control points and the more complex shape. But it is also harder to use, because each control point still influences the entire curve.

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.

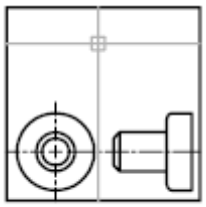
direct distance entry A method to specify a second point by first moving the cursor to indicate direction and then entering a distance.

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

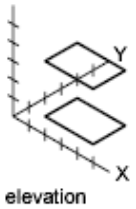
dynamic dimension Temporary dimensions that appear on objects, including dynamic block references, when they are grip edited.

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 CUIx 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 CUxI file by modifying a main CUIx 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 solid or 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 A triangular or quadrilateral portion of a 3D mesh object. Smoothing a mesh object increases the number of facets.

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 file window.

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.

G0 continuity See continuity on page 2295.

G1 continuity See continuity on page 2295.

G2 continuity See continuity on page 2295.

general property Properties that are common between a selection of objects. These include Color, Layer, Linetype, Linetype scale, Plot style, Lineweight, Transparency, Hyperlink, and Thickness.

generic surface A 3D surface object with no control vertices, history, or analytic information. Generic surfaces cannot be associative and they are created when associative analytic surfaces are separated or by using the BREP command. See also procedural surface and NURBS surface.

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.

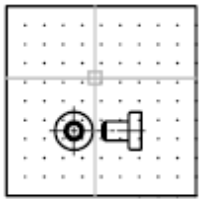
Gooch shading A type of shading that uses a transition from cool to warm colors rather than from dark to light.

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 menu options *See* multi-functional grip menu options.

grip modes The editing options you can access from selected grips on selected objects: 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)

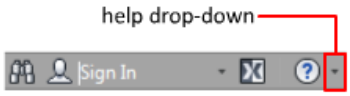
handle A unique alphanumeric tag for an object in the program's database.

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.

hot grip A selected grip.

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.dwt* or *acadiso.dwt*. *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.

isoline Gridlines that appear on regular and NURBS surfaces. They show the shape of the surface.

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.

isoparm Lines of constant *U* or *V* values that run along a NURBS surface. They show the shape of the surface as defined by the control vertices.

key point In a dynamic block definition, the point on a parameter that drives its associated action when edited in the block reference.

knot The knot parameterization setting affects the shape of a fit point spline. The different settings affect the shape of the curve as it passes through the fit points.

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.

magnitude See bulge magnitude.

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.cuix* 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 , 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-functional grip menu options Editing options you can access from the grip menu that appears when you hover over an object grip (not available for all object types).

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)

navigation bar Navigation tools that are common across multiple Autodesk programs. The unified navigation tools include Autodesk® ViewCube®, SteeringWheels®, ShowMotion®, and 3Dconnexion®.

node An object snap specification to locate points, dimension definition points, and dimension text origins.

non-associative 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.

NURBS surface Surfaces that are have control vertices in the U and V directions. NURBS surfaces cannot be associative. See also procedural surface and generic surface.

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.

periodic curve A smooth, closed curve such as a circle. Because its control vertices are not tangent to the object, if the curve is reshaped, it stays smooth and does not create kinks. See also clamp curve.

periodic surface A smooth, closed surface such as a cylinder. Because its control vertices are not tangent to the object, if the surface is reshaped, it stays smooth and does not create kinks. See also clamp surface.

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.

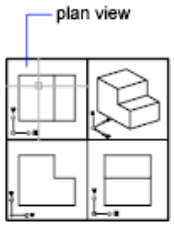
pick button The button on a pointing device that is used to select objects or specify points on the screen. For example, on a two-button mouse, it is the left button by default.

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 2314.

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 . 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*.

procedural surface A 3D surface object that has history and analytic information, but no control vertices. Procedural surfaces are the only type of surface that can be associative. See also generic surface and NURBS surface.

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, pins keep panels expanded. In the application menu, pins keep an item in the list of recently viewed 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.

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.

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, ribbon panel, *and* slide-out panel. (RIBBON)

ribbon panel A set of labeled controls, related to a task, grouped together in a ribbon. Multiple ribbon panels, belonging to one workflow, are grouped together under a ribbon tab.

ribbon tab Highest level of ribbon grouping, based on an action. A ribbon tab contains groups of multiple ribbon panels, each belonging to one workflow. A ribbon panel contains buttons and controls, related to a task.

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 anti-aliasing 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.

spline-fit Uses the vertices of the selected polyline as the control points, or frame, of a curve approximating a B-spline. This curve, called a spline-fit polyline, passes through the first and last control points unless the original polyline was closed.

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 A part of a composite object.

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subset A named collection of sheets in a sheet set that is often organized by discipline or workflow stage. *See also* view category.

surface A surface is a 3D object that is an infinitely thin shell. There are 3 types of surfaces: analytic, generic, and NURBS.

surface associativity See associative surfaces

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.

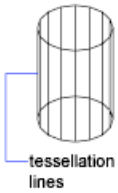
table A rectangular array of cells that contain annotation, primarily text but also blocks. In the AEC industry, tables are often referred to as "schedules" and contain information about the materials needed for the construction of the building being designed. In the manufacturing industry, they are 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 the SteeringWheels.

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.

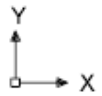
transparent command A command started while another is in progress. Precede transparent commands with an apostrophe.

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 definition Each UCS definition can have its own origin and X, Y, and Z axes. Create and save as many UCS definitions as you need.

UCS icon An icon that indicates the orientation of the UCS axes. (UCSICON)



UCS icon

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) The active coordinate system that establishes the XY plane (work plane) and Z-axis direction for drawing and modeling.

You can set the UCS origin and its X, Y, and Z axes to suit your needs. *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 non-overlapping 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 SteeringWheels that is used to organize wedges and other buttons.

wheel wedge A section on the surface of a SteeringWheels that is designated for a specific navigation or orientation tool.

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.

work plane Another name for the *XY* plane of the user coordinate system. *See also* elevation *and* user coordinate system (UCS).

working drawing A drawing for manufacturing or building purposes.

working set A group of objects selected for in-place reference editing.

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) The fixed 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)

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