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Get Information
Find the Information You Need

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

This program is a powerful application with tools that help you work with a high level of efficiency and productivity. You install this software with the Installation wizard that starts automatically when you insert the product media.

This application 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.

Find Information Using InfoCenter

You can use InfoCenter to search a variety of information sources with one query. You can also easily access product updates and announcements.

Overview of InfoCenter

You can use InfoCenter to search a variety of information sources with one query. You can also easily access product updates and announcements.

You can use InfoCenter to:

- Search for information through keywords (or by entering a phrase)
- Access subscription services through Subscription Center panel
- Access to product-related updates and announcements through Communication Center panel
Access saved topics through Favorites panel
Access topics in Help

To display the InfoCenter box in a collapsed state, click the arrow to its left.

**To browse search results**
➤ On the panel for Search Results, Subscription Center, Communication Center, or Favorites, on the right side of the category header, do one of the following:
- Click the Next button.
- Click the Previous button.

**To rearrange the topics displayed on a panel**
1 Display a panel by doing one of the following:
- In the InfoCenter box, enter a keyword or phrase. Then press ENTER or click the Search button.
- In the InfoCenter box, click the Communication Center button.
- In the InfoCenter box, click the Favorites button.

2 Click and drag a category or group header to the desired position.

**Search For Information**
You can enter keywords or a phrase in the InfoCenter box to search for information.

When you enter keywords or a phrase in the InfoCenter box, you search the contents of multiple Help resources as well as any additional documents that have been specified in the InfoCenter Settings dialog box or through the CAD Manager Control Utility.
When you enter keywords or a phrase in the InfoCenter box, you search the contents of multiple Help resources as well as any additional documents that have been specified in the InfoCenter Settings dialog box or through the CAD Manager Control Utility.

Keyword searches produce better results. In case of a misspelled word, spelling suggestions are displayed on the panel.

The results are displayed as links on the InfoCenter Search Results panel. Click a link to display the topic, article, or document.

To keep Search Results, Subscription Center, Communication Center, and the Favorites panel expanded, click the push pin icon in the bottom-right corner of the panel.

When you use InfoCenter to search for information, you can use the following special symbols in your query to refine or expand it. These symbols can be used alone or can be combined.

<table>
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<th>Symbol</th>
<th>Description</th>
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<td>*</td>
<td>Replaces one or more characters when used at the beginning, middle, or end of a word. For example, “<em>lish”, “p</em>lish”, and “pub**” will all find “publish”. Also, “anno*”</td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
</tr>
<tr>
<td>?</td>
<td>Replaces a single character. For example, “cop?” will find “copy”, but not “copybase”.</td>
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<tr>
<td>~</td>
<td>Adds grammatical form variations to a keyword when added at the beginning or end of a word. For example, “plotting~” will find “plots”, “plotted”, and so on. Also, “~plot” will find “preplot”, “replot”, and so on.</td>
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When performing the exact phrase search, use double quotation marks (" ") to enclose words that must appear next to each other in the specified sequence. For example, enter "**specify units of measurement**" to find only topics with all those words in that order. You can also use the previously mentioned symbols in a text string that is enclosed in double quotation marks.

**See also:**
- Specify InfoCenter Settings on page 11

**To search multiple sources for information**
1. In the InfoCenter box, enter a keyword or phrase.
2. Click the Search button.

**To search a single location for information**
1. In the InfoCenter box, enter a keyword or phrase.
2. Click the down arrow next to the Search button.
3. Select a location from the list to search.

**To add a location to search**
1. In the InfoCenter box, click the down arrow next to the Search button.
2. Click Add Search Location.
3. In the Add Search Location dialog box, specify a document or a file location to search.
4. Click Add.
Access Subscription Center

You can click the Subscription Center button to display links to information about subscription services such as product enhancements, personalized web support from Autodesk technical experts, and self-paced e-Learning. To learn more about Autodesk subscription membership, visit http://www.autodesk.com/subscriptioncenter.

Autodesk Subscription Services

With Autodesk Subscription, you get the latest releases of Autodesk software, incremental product enhancements, personalized web support from Autodesk technical experts, and self-paced e-Learning.

Subscription services are available to subscription members only (available in countries/regions where Autodesk subscriptions are offered).

Click the Subscription Center button in the InfoCenter box to access the following options:

- **Create support request.** Provides direct communication with Autodesk support technicians. You receive fast, complete answers to your installation, configuration, and troubleshooting questions.

- **View support request.** Tracks and manages your questions and responses through Autodesk's support system.

- **Edit Subscription Center profile.** Sets up and maintains your subscription membership.

- **View e-Learning catalog.** Features interactive lessons organized into product catalogs.

- **e-Learning lessons.** Provides e-Learning lessons. Each lesson is 15-30 minutes and features hands-on exercises, with an option to use a simulation instead of the software application. You can use an online evaluation tool that identifies gaps in skills, determines the lessons that will be most helpful, and keeps track of learning progress.

Subscription Resources and Privacy

Subscription resources provide interactive product features over the Internet. Each time you access subscription resources from Subscription Center in an
Autodesk product, product information (such as the serial number, version, language, and the subscription contract ID) is sent to Autodesk for verification that your product is on subscription.

Autodesk compiles statistics using the information sent to subscription resources to monitor how they are being used and how they can be improved. Autodesk maintains the information provided by or collected from you in accordance with Autodesk’s published privacy policy, which is available at http://www.autodesk.com/privacy.

Enable Subscription Resources
The CAD Manager Control Utility can be used to turn Subscription Resources on and off in the program.

To access the Subscription
1. In the InfoCenter box, click the Subscription Center button.
2. On the Subscription Center panel, under Subscription Center, click the subscription resource you want to access.

NOTE Subscription Center is not available to all product users. If subscription resources are not available in your product, your product is not entitled to subscription benefits.

Receive Product Updates and Announcements
You can click the Communication Center button to display links to information about product updates and announcements, and may include links to RSS feeds.

Overview of Communication Center
To open Communication Center, click the Communication Center button in the InfoCenter box.
Communication Center provides the following types of announcements:

- **Autodesk Channels**: Receive support information, product updates, and other announcements (including articles and tips).

- **CAD Manager Channel**: Receive information (RSS feeds) published by your CAD manager.

- **RSS Feeds**: Receive information from RSS feeds to which you subscribe. RSS feeds generally notify you when new content is posted. You are automatically subscribed to several default RSS feeds when you install the program.

You can customize the items that display in the Communication Center panel.

**Communication Center Online Privacy**

Communication Center is an interactive feature that must be connected to the Internet in order to deliver content and information. Each time Communication Center is connected, it sends your information to Autodesk so that you receive the correct information. All information is sent anonymously to Autodesk to maintain your privacy.

The following information is sent to Autodesk:

- Product name (in which you are using Communication Center)
- Product release number
- Product language
- Country/region (specified in the Communication Center settings)
- Your subscription contract number (if you’re a subscription customer)

Autodesk compiles statistics using the information sent from Communication Center to monitor how it is being used and how it can be improved. Autodesk maintains information provided by or collected from you in accordance with the company’s published privacy policy, which is available on [http://www.autodesk.com/privacy](http://www.autodesk.com/privacy).

**See also:**

- Specify InfoCenter Settings on page 11
- Access Subscription Center on page 7
Receive New Information Notifications

Whenever new information is available, Communication Center notifies you by displaying a balloon message below the Communication Center button on the InfoCenter box.

Click the link in the balloon message to open the article or announcement.

If you don’t want to receive Communication Center notifications, in the InfoCenter Settings dialog box, turn off Balloon Notification.

See also:

- Specify InfoCenter Settings on page 11

Save and Access Favorite Topics

You can click the Favorites button to display saved links to topics or web locations.

Any link that displays on the Search Results panel, Subscription Center or Communication Center panel can be marked as a favorite.

A link marked as a favorite displays a star icon on the Search Results panel, Subscription Center panel or the Communication Center panel.

To display the InfoCenter Favorites panel

- In the InfoCenter box, click the Favorites button.

To save a link in InfoCenter as a favorite

1 Display a panel by doing one of the following:
   - In the InfoCenter box, enter a keyword or phrase. Then press ENTER or click the Search button.
   - In the InfoCenter box, click the Subscription Center button.
   - In the InfoCenter box, click the Communication Center button.

2 Click the star icon that is displayed next to the link that you want to save as a favorite.
To remove a favorite link from the InfoCenter Favorites panel

1. In the InfoCenter box, click the Favorites button to display the Favorites panel.

2. Click the star icon that is displayed next to the link that you want to remove from the Favorites panel.

Specify InfoCenter Settings

You can specify InfoCenter Search and Communication Center settings in the InfoCenter Settings dialog box.

In the InfoCenter Settings dialog box, you can specify the following settings:

- **General.** Your current location, frequency for checking new online content and option to turn on or off animated transition effects for the InfoCenter panels.

- **Search Locations.** Locations (documents, web locations, and files) to search for information, as well as the name that displays for each location and the number of results to display for each. Also, you can add or remove search locations.
  
  The Web Locations check box provides access to important information on the Autodesk website, including the Knowledge Base and discussion groups. When you add document locations, you can specify files on your local drive.

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**NOTE** User-specified CHM (compiled help) files must be located on your local drive. InfoCenter cannot search CHM files located on network drives.
Communication Center. Maximum age of the articles displayed on the Communication Center panel and the location and name of the CAD Manager Channel.

Autodesk Channels. Channels to display in the Communication Center panel as well as the number of articles to display for each channel.

Balloon Notification. Notifications for new product information, software updates, product support announcements, and Did You Know messages. Also, you can customize the transparency and the display time of the balloon.

NOTE Did You Know messages displayed below the Communication Center button on the InfoCenter box provide Knowledge Base information and general instructional messages such as tips. You can click on the text or the Expand icon to expand the balloon to view the detailed information.

RSS Feeds. RSS feed subscriptions. You can add or remove RSS feeds. RSS feeds generally notify you when new content is posted.

You can use the CAD Manager Control utility to specify InfoCenter Search and Communication Center settings. For more information about how to install and run the utility, see To install the CAD Manager Control utility on page 102. Click Help in the CAD Manager Control utility window for details about the settings you can control.

To specify locations to search for information

1. In the InfoCenter box, click the down arrow next to the Search button.
2. Click Search Settings.
3. In the InfoCenter Settings dialog box, Search Locations panel, in the right pane, select or clear the search locations you want to include or exclude when you search for information.
4. Click OK.

To add new location to search for information

1. In the InfoCenter box, click the down arrow next to the Search button.
2. Click Search Settings.
3. In the InfoCenter Settings dialog box, do one of the following:
   - On the Search Locations panel, in the right pane, click Add.
On the Search Locations panel, in the right pane, right-click anywhere in the pane. Click Add.

4 In the Add Search Location dialog box, specify a file location to search.
5 Click Add.
6 Click OK.

To remove a search location
1 In the InfoCenter box, click the down arrow next to the Search button.
2 Click Search Settings.
3 In the InfoCenter Settings dialog box, do one of the following:
   ■ Select a location to remove, and then click Remove.
   ■ Right-click a search location. Click Remove.
4 In the InfoCenter - Remove Search Location dialog box, click Yes.
5 Click OK.

To specify the CAD Manager Channel location and name
1 Click Start menu (Windows) ➤ All Programs (or Programs) ➤ Autodesk ➤ CAD Manager Tools ➤ CAD Manager Control Utility.
2 Select the product you want to modify. Click OK.
3 In the CAD Manager Control Utility window, InfoCenter tab, select the option to enable CAD Manager Channel.
4 Enter an appropriate feed under Feed Location.
5 Enter the display name for the CAD Manager and then click Apply. Click OK.
6 Restart AutoCAD.
7 In the upper right-side of the application, on the InfoCenter box, click the arrow next to the Search button.
8 Click Search Settings.
In the InfoCenter Settings dialog box, in the left pane, click Communication Center. The CAD Manager Channel location and name are displayed.

**NOTE** A green check mark is displayed if the channel can be located and a yellow warning sign is displayed if it cannot be located.

### To specify the channels to display in the Communication Center panel

1. In the InfoCenter box, click the down arrow next to the Search button.
2. Click Search Settings.
3. In the InfoCenter Settings dialog box, in the left pane, click Autodesk Channels.
4. In the right pane, select or clear the channels you want to display in the Communication Center panel.
5. Click OK.

### To specify InfoCenter balloon notification settings

1. In the InfoCenter box, click the arrow next to the Search button.
2. Click Search Settings.
3. In the InfoCenter Settings dialog box, in the left pane, click Balloon Notification.
4. In the right pane, select or clear the options to turn balloon notification on or off.

**NOTE** You can turn off the balloon notification for Did You Know messages in the InfoCenter Settings dialog box.

5. Select one of the following options to set the display time for the Did You Know messages:
   - **Use Balloon Notification Display Time.** Uses the display time specified in the InfoCenter Settings dialog box (Balloon Notification panel) for the Did You Know messages. This option is unavailable if the Did You Know messages option is cleared.
   - **Display until Closed.** Displays the Did You Know message until you click the Close button.
6 Enter the number of seconds to set the length of time for balloon notifications to display.

7 Enter the transparency value of the balloon or set the value using the slider.

8 Click OK.

**To turn on the display of Did You Know hidden messages**

1 Click Tools menu ➤ Options.

2 In the Options dialog box, System tab, under General Options, click the Hidden Message Settings button.

3 In the Hidden Message Settings dialog box, select the InfoCenter Did You Know Balloons option to turn on the display of all hidden messages. Click OK.

4 Click OK.

**To add an RSS feed to Communication Center**

1 In the InfoCenter box, click the down arrow next to the Search button.

2 Click Search Settings.

3 In the InfoCenter Settings dialog box, in the left pane, click RSS Feeds.

4 In the right pane, do one of the following:
   - Click Add.
   - Right-click anywhere in the right pane. Click Add.

5 In the Add RSS Feed dialog box, enter the location of the RSS feed you want to add. Click Add.

6 In the InfoCenter - RSS Feed Confirmation dialog box, click Close.

7 Click OK.

**To remove an RSS feed from Communication Center**

1 In the InfoCenter box, click the down arrow next to the Search button.

2 Click Search Settings.

3 In the InfoCenter Settings dialog box, in the left pane, click RSS Feeds.
4 In the right pane, do one of the following:
   ■ Click Remove.
   ■ Right-click an RSS feed. Click Remove.

5 In the InfoCenter - Remove RSS Feed dialog box, click Yes.
6 Click OK.

Search Topics in Help
You can click the Help button to display topics in Help.

![Help button]

You can get much more benefit from the Help system when you learn how to use it efficiently. You can quickly find general descriptions, procedures, details about dialog boxes and palettes, or definitions of terms.

To display topics in Help
   ■ In the InfoCenter box, click the Help button.

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

- New Features Workshop
  The New Features Workshop introduces you to what’s new in PRODNAME.

- Authorized Training Centers
  More than 1,200 ATC sites are available worldwide to meet your needs for discipline-specific, locally based training.
Autodesk Official Training Courseware

Autodesk Official Training Courseware (AOTC) is technical training material developed by Autodesk. You can purchase AOTC from your local reseller or distributor, or you can order it online from the Autodesk Store.

e-Learning

Autodesk e-Learning for Autodesk Subscription customers features interactive lessons organized into product catalogs.

Autodesk Developer Network

The Autodesk Developer (ADN) program provides support for full-time, professional developers who want to build software based on Autodesk products.

Consulting

Autodesk Consulting provides services that help set up processes and provide critical training that will help increase productivity so you can capitalize on the power of your products.

Partner Products and Services

Visit the Partner Products & Services page for a list of resources available for your Autodesk product and your industry.

View the Product Readme

You can find late-breaking information about this software in the Readme. It is suggested that you read through the Readme for information about recommended hardware, updated installation instructions, and known software problems.

■ View the Readme
Join the Customer Involvement Program

You are invited to help guide the direction of Autodesk design software.

If you participate in the Customer Involvement Program (CIP), specific information about how you use AutoCAD LT 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 LT.

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

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.

This information includes the following:

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

This information can help you document a drawing and provides the total amount of time spent in the drawing file.

See also:

- Enter System Variables on the Command Line on page 54
- Add Identifying Information to Drawings on page 158
- Extract Geometric Information from Objects on page 447
- Compare Dimension Styles and Variables on page 963
Quick Reference

Commands

DWGPROPS
Sets and displays the file properties of the current drawing.

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

**The Application Menu**

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

**Search for Commands**

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

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

**NOTE** You can also close AutoCAD LT 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.

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**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 differs 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 141
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.

**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.

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

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.

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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.
Other Tool Locations

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 LT 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 141
- “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

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 LT® 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 32.

NOTE You can turn a toolbar into a ribbon panel using the Customize User Interface dialog box.
Display or Hide, Dock, and Resize Toolbars

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

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

See also:

■ Create Task-Based Workspaces on page 141
■ “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. The lock button locks the current positions of the toolbars and windows. To expand the drawing display area, click the Clean Screen button.

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

NOTE When the application status bar is turned off, the Clean Screen button is not displayed on the screen.
To control the display of icons and notifications in the status bar tray

1. Click View tab ➤ Windows panel ➤ Status Bar.
2. In the Status Bar drop-down, click Tray Settings.
3. In the Tray Settings dialog box, select or clear the following display options:
   ■ Display Icons from Services. Displays the tray at the right end of the status bar and displays icons from services. When this option is cleared, the tray is not displayed.
   ■ Display Notifications from Services. Displays notifications from services such as Communications Center. When the Display Icons from Services option is cleared, this option is unavailable.
4. If Display Notifications from Services is selected, set a time for a notification to be displayed, or select Display Until Closed.
5. Click OK.

To control the display of buttons on the status bar

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

To control the display of cursor coordinates on the status bar

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

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.

Drawing Status Bar

The drawing status bar displays several tools for scaling annotations. Different tools display for model space and paper space.

When the drawing status bar is turned on, it displays at the bottom of the drawing area. When the drawing status bar is turned off, the tools found on the drawing status bar are moved to the application status bar.

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

To turn the drawing status bar on or off

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

To control the display of buttons on the drawing status bar

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

Quick Reference

System Variables

STATUSBAR

Controls the display of the application and drawing status bars.
Keytips

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

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

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

The Command Window

Enter Commands on the Command Line

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

Commands, system variables, options, messages, and prompts are displayed in a dockable and resizable window called the *command window*. The bottom line of the command window is called the *command line*. The command line displays the operation in progress and provides an inside view of exactly what the program is doing.
To enter a command by using the keyboard, type the full command name on the command line and press Enter or Spacebar.

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

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

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

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

**Specify Command Options**

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

Specify center point for circle or [3P/2P/Ttr (tan, tan, radius)]:
You can specify the center point either by entering $X,Y$ coordinate values or by using the pointing device to click a point on the screen.

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

### Execute Commands

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

### Repeat and Cancel Commands

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

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

Command: multiple circle

To cancel a command in progress, press ESC.

### Interrupt a Command with Another Command or System Variable

Many commands can be used transparently: that is, they can be entered on the command line while you use another command. Transparent commands frequently change drawing settings or display, for example, GRID or ZOOM. In the Command Reference, transparent commands are designated by an apostrophe in front of the command name.

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

Command: line

Specify first point: 'grid

>>Specify grid spacing (X) or [ON/OFF/Snap/Aspect] <0.000>: 1

Resuming LINE command

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

See also:

- “Keyboard Shortcuts” in the Customization Guide

To copy a command you have recently used

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

Quick Reference

MULTIPLE
Repeats the next command until canceled.

OPTIONS
Customizes the program settings.

PASTECLIP
Paste objects from the Clipboard into the current drawing.

System Variables

CMDNAMES
Displays the names of the active and transparent commands.

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:

<table>
<thead>
<tr>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Toolbars and windows not locked</td>
</tr>
<tr>
<td>1</td>
<td>Docked toolbars locked</td>
</tr>
<tr>
<td>2</td>
<td>Docked or anchored windows locked</td>
</tr>
<tr>
<td>4</td>
<td>Floating toolbars locked</td>
</tr>
<tr>
<td>8</td>
<td>Floating windows locked</td>
</tr>
</tbody>
</table>

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 60
- Use Dynamic Input on page 406

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 text window log file opened by LOGFILEON.

LOGFILEON
Writes the text window contents to a file.

PASTECLIP
Pastes objects from the Clipboard into the current drawing.

TEXTSCR
Opens the text window.

System Variables

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

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

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

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 LT® 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 LT 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 LT window.

Anchoring the Command Window

The command window can be anchored on the left or right side of the AutoCAD LT 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 107.

**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 LT 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 LT 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 acadlt.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
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 display of a history of user input.

PICKFIRST
Controls whether you select objects before (noun-verb selection) or after you issue a command.
SHORTCUTMENU
Controls whether Default, Edit, and Command mode shortcut menus are available in the drawing area.

TRAYICONS
Controls whether a tray is displayed on the status bar.

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

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

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
- Raster images
- External references (xrefs)
NOTE When you drag an object onto a tool palette, you can switch to a different tab by hovering over the tab for a few seconds.

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

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

Insert Blocks and Attach References

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

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

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

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

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

If the Redefine option is unavailable, then the block definition source is a drawing file rather than a block within a drawing file. To update a block definition that was created by inserting a drawing file, use DesignCenter. For more information, see Add Content with DesignCenter on page 95.

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 76
- Customize Tool Palettes on page 81
- Create Task-Based Workspaces on page 141
- Add Content with DesignCenter on page 95

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 tool or the dimension tool whose flyout you want to add or remove. Click Properties.

2. In the Tool Properties dialog box, under Command, click in the Use Flyout box.
3 In the drop-down list, select Yes if you want to add a flyout, or select No if you want to remove one.

4 Click OK.

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

---

### To customize a tool flyout

1 On a tool palette, right-click the geometric object tool or dimension tool whose flyout you want to customize. Click Properties.

2 In the Tool Properties dialog box, under Command, click in the Flyout Options box. Click the [...] button.

3 In the Flyout Options dialog box, select the tools that you want to show on the flyout. Click OK. (You must select at least one tool.)

4 In the Tool Properties dialog box, click OK.

---

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

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

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

3 In the drop-down list, select Yes.

4 Click OK.

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

INSUNITSDefsSOURCE
  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 to a tool palette or you can drag tools from the Customize User Interface (CUI) Editor to a tool palette.

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

You can also create a tool that executes a string of commands or a script.
To create a command tool from a toolbar button

1. Make sure the toolbar that contains the command you want to add to the tool palette is displayed.
   - If the required toolbar is not displayed, click Tools ➤ Toolbars and select another toolbar from the list.

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

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

3. In the program, drag a command (button) from a toolbar to the tool palette and, without releasing the mouse button, move the cursor to the place on the tool palette where you want the tool.
   - The black horizontal line indicates where the tool will be located.

4. Release the mouse button.

5. In the Customize dialog box, click Close.

To create a command tool from the Customize User Interface

1. Click Manage tab ➤ Customization panel ➤ User Interface.
   - If the CUI Editor is covering the Tool Palettes window, then move the CUI Editor to the side.

2. In the Command List pane, drag a command to the tool palette and, without releasing the mouse button, move the cursor to the place on the tool palette where you want the tool.
   - The black horizontal line indicates where the tool will be located.

3. Release the mouse button.

4. In the CUI Editor, click Close.
To create a command tool that executes a string of commands or a script (advanced)

1. Click Manage tab ➤ Customization panel ➤ Tool Palettes.
2. In the program, drag a command from a toolbar to the tool palette and, without releasing the mouse button, move the cursor to the place on the tool palette where you want the tool.
3. Release the mouse button.
4. On the tool palette, right-click the tool. Click Properties.
5. In the Tool Properties dialog box, change the name and description to an appropriate name and description for the string or script.
6. Under Command, in the Command String box, enter a string of commands or a script.
7. Click OK.

To use a command tool
1. On a tool palette, click the command tool that you want to use.
2. Follow any Command prompts that are shown.

Quick Reference

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Change Tool Palette Settings

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

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

These settings include

- **Allow Docking**. Toggles the ability to dock or anchor palette windows. If this option is selected, a window can be docked when you drag it over a docking area at the side of a drawing. A docked window adheres to the side of the application window and causes the drawing area to be resized. Selecting this option also makes Anchor Right and Anchor Left available.

- **Anchor Left** or **Anchor Right**. Attaches the palette to an anchor tab base at the left or right side of the drawing area. The palette rolls open and closed as the cursor moves across it. When an anchored palette is open, its content overlaps the drawing area. An anchored palette cannot be set to stay open.

- **Auto-hide**. Controls the display of the palette when it is floating. When this option is selected, only the tool palette title bar is displayed when the cursor moves outside the tool palette. When this option is cleared, the palette stays open continuously. You can display the tool palette title bar as either icons or text from the shortcut menu of the title bar.

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

- **Views**. Changes the display style and size of the icons in a tool palette.
To change the rollover behavior of the Tool Palettes window

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

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

To change the transparency of the Tool Palettes window

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

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

To change the icon display style in the Tool Palettes window

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

**Quick Reference**

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.

**Control Tool Properties**

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

Once a tool is on a tool palette, you can change its properties. For example, you can change the insertion scale of a block or the angle of a hatch pattern.
To change tool properties, right-click on a tool, and then click Properties on the shortcut menu to display the Tool Properties dialog box. The Tool Properties dialog box has the following two categories of properties:

- **Insert or Pattern properties.** Control object-specific properties such as scale, rotation, and angle.

- **General properties.** Override the current drawing property settings such as layer, color, and linetype.

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

**Specify a Different Icon for a Tool**

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

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

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

**Update the Icon for a Tool**

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

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

**Specify Overrides for Tool Properties**

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

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

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

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

**To display the properties of a tool on a tool palette**

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

2. In the Tool Properties dialog box, use the scroll bar to view all tool properties. You can resize the Tool Properties dialog box by dragging an edge, or you can expand and collapse the property categories by clicking the double arrow buttons.

3. Click OK.

**To change the property of a tool on a tool palette**

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

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

- Properties listed under the Insert or Pattern category control object-specific properties such as scale, rotation, and angle.
Properties listed under the General category override the current drawing property settings such as layer, color, and linetype.

Auxiliary scale for a block or a hatch tool overrides the regular scale setting when the tool is used. (An auxiliary scale multiplies your current scale setting by the plot scale or the dimension scale.)

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

3 Click OK.

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

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**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.

2 On the shortcut menu, click Specify Image.

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

4 Click Open to insert the new image.

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

1. On a tool palette, right-click a tool. Click Properties.
2. In the Tool Properties dialog box, right-click in the image area. Click Delete Image.
3. Click in the Name box and delete the text.
4. Click in the Description box and delete the text.
5. Click OK.

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

**Quick Reference**

CUSTOMIZE
- Customizes tool palettes and tool palette groups.

TOOLPALETTES
- Opens the Tool Palettes window.

TOOLPALETTESTCLOSE
- 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. 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; blocks; xrefs.

■ Drag drawings, blocks, and hatches from DesignCenter to the tool palette. Drawings that are added to a tool palette are inserted as blocks when dragged into the drawing.

■ Use the Customize dialog box to drag commands to a tool palette just as you might add them to a toolbar.

■ Use the Customize User Interface (CUI) Editor to drag commands to a tool palette from the Command List pane.

■ Use Cut, Copy, and Paste to move or copy tools from one tool palette to another.

■ Manage tool palettes by creating new palettes from scratch, renaming, deleting or moving palettes with the shortcut menu.

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

■ Create a tool palette tab with predetermined content by right-clicking a folder, a drawing file, or a block in the DesignCenter tree view, and then clicking Create Tool Palette on the shortcut menu.
You can associate a customizable tool palette group with each panel on the ribbon. Right-click the ribbon panel to display a list of available tool palette groups.

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

**Rearranging Tools and Tool Palettes**

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

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

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

To apply a read-only attribute to a tool palette, right-click the tool palette (ATC) file in the following location: C:\documents and settings\<user name>\application data\autodesk\AutoCAD LT 2011\r17.2\enu\support\ToolPalette\Palettes. On the shortcut menu, click Properties. On the General tab, select Read-only, and click OK.

To create a tool palette

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

To associate a tool palette group with a ribbon panel

1. Click View tab ➤ Palettes panel ➤ Tool Palettes.
2. On the ribbon, right-click a ribbon panel and click Tool Palette Group.
3. Click an available tool palette group from the list.
4. Click Tools menu ➤ Workspaces ➤ Save Current As.
5. In the Save Workspace dialog box, enter a name for the new workspace or select a name from the drop-down list. Click Save.

To display the tool palette group associated with a ribbon panel

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

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

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

When you set the Hatch Pattern group as the current group, only those tool palettes you've added to the group are displayed.
To create or remove a tool palette group

1 Click Manage tab ➤ Customization panel ➤ Tool Palettes.

2 In the Customize dialog box, under Palette Groups, right-click on the lower, blank area. Click New Group.
   If there are no groups listed in the Palette Groups area, you can create a group by dragging a tool palette from the Tool Palettes area into the Palette Groups area.

3 Enter a name for the tool palette group.

4 Click Close.

**NOTE** To delete a tool palette group, in the Customize dialog box, under Palette Groups, right-click the tool palette group you want to delete. Click Delete.

To add a tool palette to a tool palette group

1 Click Manage tab ➤ Customization panel ➤ Tool Palettes.

2 In the Customize dialog box, drag a tool palette from the Tool Palettes area into a group in the Palette Groups area.

3 Click Close.

To display a tool palette group

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

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

To delete all tool palette groups

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

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

2 Click Manage tab ➤ Customization panel ➤ Tool Palettes.
In the Customize dialog box, under Palette Groups, right-click a tool palette group. Click Delete.

Repeat step 3 until all tool palette groups are deleted.

Click Close.

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

1. Click Manage tab ➤ Customization panel ➤ Tool Palettes.
2. In the Customize dialog box, under Palette Groups, select the tool palette that you want to copy.
3. Press Ctrl while you drag the selected tool palette to another group. When you release the mouse button, a copy of the tool palette is displayed in the new location.
4. Click Close.

To display all tool palettes
Right-click on the title bar of a tool palette. Click All Palettes.

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

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

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

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

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

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

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

Tool palettes can be used only in the version of AutoCAD LT in which they were created. For example, you cannot use a tool palette that was created in AutoCAD LT 2011 in AutoCAD 2005.

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

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

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

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

1. Click Manage tab ➤ Customization panel ➤ Tool Palettes.
2. In the Customize dialog box, under Palettes, right-click a tool palette. On the shortcut menu, click Export.
3. In the Export Palette dialog box, enter a file name and click Save.
4. Click Close.

To share a tool palette group

1. Click Manage tab ➤ Customization panel ➤ Tool Palettes.
2. In the Customize dialog box, under Palette Groups, right-click a tool palette group. Click Export.
3. In the Export Group dialog box, enter a file name and click Save.
4. Click Close.

Quick Reference

CUSTOMIZE
Customizes tool palettes and tool palette groups.

TOOLPALETTES
Opens the Tool Palettes window.

TOOLPALETTESCLOSE
Closes the Tool Palettes window.

DesignCenter

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

**Overview of DesignCenter**

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

**Quick Reference**

**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.
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.

Open Drawings, History, and DC Online Tabs

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

- **Open Drawings.** Displays a list of the drawings that are currently open. Click a drawing file and then click one of the definition tables from the list to load the content into the content area.

- **History.** Displays a list of files opened previously with DesignCenter. Double-click a drawing file from the list to navigate to the drawing file in the tree view of the Folders tab and to load the content into the content area.
- **DC Online.** Provides content from the DesignCenter Online web page including blocks, symbol libraries, manufacturer's content, and online catalogs.

**NOTE** The DesignCenter Online (DC Online tab) is disabled by default. You can enable it from the CAD Manager Control utility on page 102.

**Bookmark Frequently Used Content**

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

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

**To change the source of the content displayed in DesignCenter**

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

**NOTE** The DesignCenter Online (DC Online tab) is disabled by default. You can enable it from the CAD Manager Control utility on page 102.

**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
Loading a specified DesignCenter drawing file, folder, or network path.

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.

NOTE  The DesignCenter Online (DC Online tab) is disabled by default. You can enable it from the CAD Manager Control utility on page 102.

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 102.

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](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 Review the Autodesk software license agreement for your country or region. You must accept this agreement to proceed 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.

2 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.

3 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 102.

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 1366

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

**Set Interface Options**

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

**Set Up the Drawing Area**

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

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

Some settings affect how you work in the drawing area:

- **Background Colors (Options dialog box, Display tab)**. You specify the background colors used in the layout and Model tabs and the color used for prompts and crosshairs.
- **Color Scheme (Options dialog box, Display tab, Colors).** You specify a dark or light color scheme for the overall user interface. The settings affect the window frame background, status bar, title bar, menu browser frame, toolbars, and palettes.

- **Background Colors (Options dialog box, Display tab, Colors).** You specify the background colors used in model space, layouts, and the block editor. Background colors on the Model tab change to indicate whether you are working in a 2D design context, 3D modeling (parallel projection), or 3D modeling (perspective projection).

- **UCS Icon and Crosshairs Cursor (Options dialog box, 3D Modeling tab).** You specify that the 3D display options and labels for the UCS icon can be set in the 3D Modeling tab of the Options dialog box.

- **Color Assignments for X, Y, and Z (Options dialog box, Display tab, Colors).** In 3D views, any interface elements that are associated with the UCS X, Y, and Z axis use special color assignments. The X axis is colored or tinted red, the Y axis is green, and the Z axis is blue. These tints can be turned on or off in the Drawing Window Colors dialog box.

- **Clean Screen.** You can expand the drawing display area to display only the menu bar, status bar, and command window with the clean screen button on the application status bar. Click the button again to restore the previous setup.

- **View Transitions.** You can control whether view transitions are smooth or instantaneous when you pan, zoom, or change from one view to another (VTOPTIONS command). The default is a smooth transition.

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

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

**See also:**
- Display Tab (Options Dialog Box) in the Command Reference
- User Interface Customization in the Customization Guide
Rollover Tooltips in the Customization Guide

Create Tooltips and Extended Help for Commands in the Customization Guide

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

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 dialogs**

1 Click the Application button. At the bottom of the Application menu, click Options.

2 In the Options dialog box, System tab, under General Options, click Hidden Messages Settings button.

3 The Hidden Message Settings dialog box is displayed.

4 Check the corresponding dialog name from the tree directory.

5 Click OK.

**To turn tooltips on or off**

1 Click Tools menu ➤ Options.

2 In the Options dialog box, Display tab, under Window Elements, click Show Tooltips.

3 Click OK.
Quick Reference

Commands
CLEANSCREENON
Clears the screen of toolbars and dockable windows, excluding the command window.
CLEANSCREENOFF
Restores display of toolbars and dockable windows, excluding the command window.
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 way dragged objects are displayed.
EXEDIR
Displays the folder path of the AutoCAD LT executable file.
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.

LOCALE
Displays a code that indicates the current locale.
LOCALROOTPREFIX
Stores the full path to the root folder where local customizable files were installed.

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

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

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

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

MTEXTED
Sets the application for editing multiline text objects.

OLEQUALITY
Sets the default plot quality for OLE objects.

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

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

PAPERUPDATE
Controls the display of a warning dialog 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 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.
PICKADD
  Controls whether subsequent selections replace the current selection set or add to it.

PICKAUTO
  Controls automatic windowing at the Select Objects prompt.

PICKBOX
  Sets the object selection target height, in pixels.

PICKDRAG
  Controls the method of drawing a selection window.

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

PICKSTYLE
  Controls the use of group selection and associative hatch selection.

PSTYLEPOLICY
  Controls the plot style mode, Color-Dependent or Named, that is used when opening a drawing that was created in a release prior to AutoCAD 2000 or when creating a new drawing from scratch without using a drawing template.

QTEXTMODE
  Controls how text is displayed.

RASTERPREVIEW
  Controls whether BMP preview images are saved with the drawing.

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

ROLLOVERTIPS
  Controls the display of rollover tooltips in the application.

SAVEFILE
  Stores the current automatic save file name.
SAVEFILEPATH
Specifies the path to the directory for all automatic save files for the current session.

SAVETIME
Sets the automatic save interval, in minutes.

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

TDUSRTIMER
Stores the user-elapsed timer.

TOOLTIPMERGE
Combines drafting tooltips into a single tooltip.

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

USERNAME
Specifies the user name.

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 Space and Layouts

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

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

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

To change the Model and layout tabs to status bar buttons
- Right-click the Model tab or a layout tab. Click Hide Layout and Model Tabs.

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

Quick Reference

Commands

DRAGMODE
Controls the way dragged objects are displayed.

OPTIONS
Customizes the program settings.

VIEWRES
Sets the resolution for objects in the current viewport.

System Variables

APERTURE
Sets the display size for the object snap target box, in pixels.
CALCINPUT
  Controls whether mathematical expressions and global constants are evaluated in text and numeric entry boxes of windows and dialog boxes.

CLEANSCREENSTATE
  Indicates whether the clean screen state is on or off.

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

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

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

DEFPLSTYLE
  Specifies the default plot style for new objects in a drawing when opening a drawing that was created in a release prior to AutoCAD 2000, or when creating a new drawing from scratch without using a drawing template.

DRAGMODE
  Controls the way dragged objects are displayed.

EXEDIR
  Displays the folder path of the AutoCAD LT executable file.

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.

LOCALE
Displays a code that indicates the current locale.

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

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

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

LOGFILENAME
Specifies the path and name of the text window log file for the current drawing.
LOGFILEPATH
Specifies the path for the text window log files for all drawings in a session.

MTEXTED
Sets the application for editing multiline text objects.

OLEQUALITY
Sets the default plot quality for OLE objects.

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

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

PAPERUPDATE
Controls the display of a warning dialog 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 at the Select Objects prompt.

PICKBOX
Sets the object selection target height, in pixels.

PICKDRAG
Controls the method of drawing a selection window.

PICKFIRST
Controls whether you select objects before (noun-verb selection) or after you issue a command.
PICKSTYLE
Controls the use of group selection and associative hatch selection.

PSTYLEPOLICY
Controls the plot style mode, Color-Dependent or Named, that is used when opening a drawing that was created in a release prior to AutoCAD 2000 or when creating a new drawing from scratch without using a drawing template.

QTEXTMODE
Controls how text is displayed.

RASTERPREVIEW
Controls whether BMP preview images are saved with the drawing.

ROLOVERTIPS
Controls the display of rollover tooltips in the application.

SAVEFILEPATH
Stores the current automatic save file name.

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

SAVETIME
Sets the automatic save interval, in minutes.

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

TDUSRTIMER
Stores the user-elapsed timer.

TOOLTIPMERGE
Combines drafting tooltips into a single tooltip.

TOOLTIPS
Controls the display of tooltips on the ribbon, toolbars, and other user interface elements.
USERNAME
  Specifies the user name.

VISRETAI N
  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 Options dialog box, Display
tab.

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

To change the font displayed in the Command window

1 Click the Application button. At the bottom of the Application menu,
click Options.

2 In the Options dialog box, Display tab, under Window Elements, click
Fonts.

3 In the Command Line Window Font dialog box, select the appropriate
Font, Font Style, and Size.
   An example of the current choices appears under Sample Command Line
   Font.

4 Click Apply & Close to record the current option settings in the system
registry and close the dialog box.
5. In the Options dialog box, click OK.

**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.

EXEDIR
- Displays the folder path of the AutoCAD LT executable file.
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.

LOCALE
Displays a code that indicates the current locale.
LOCALROOTPREFIX
Stores the full path to the root folder where local customizable files were installed.

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

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

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

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

MTEXTED
Sets the application for editing multiline text objects.

OLEQUALITY
Sets the default plot quality for OLE objects.

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

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

PAPERUPDATE
Controls the display of a warning dialog 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 at the Select Objects prompt.

PICKBOX
Sets the object selection target height, in pixels.

PICKDRAG
Controls the method of drawing a selection window.

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

PICKSTYLE
Controls the use of group selection and associative hatch selection.

PSTYLEPOLICY
Controls the plot style mode, Color-Dependent or Named, that is used when opening a drawing that was created in a release prior to AutoCAD 2000 or when creating a new drawing from scratch without using a drawing template.

QTEXTMODE
Controls how text is displayed.

RASTERPREVIEW
Controls whether BMP preview images are saved with the drawing.

ROLLOVERTIPS
Controls the display of rollover tooltips in the application.

SAVEFILEPATH
Stores the current automatic save file name.

SAVEFILEPATH
Specifies the path to the directory for all automatic save files for the current session.
SAVETIME
Sets the automatic save interval, in minutes.

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

TDUSRTIMER
Stores the user-elapsed timer.

TOOLTIPMERGE
Combines drafting tooltips into a single tooltip.

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

USERNAME
Specifies the user name.

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 Properties 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 text window log file opened by LOGFILEON.

LOGFILEON
Writes the text window contents to a file.

OPTIONS
Customizes the program settings.

REVDATE
Inserts or updates a block containing user name, current time and date, and drawing name.

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.

EXEDIR
Displays the folder path of the AutoCAD LT executable file.

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.

INSUNITSDlabsource
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.

LOCALE
Displays a code that indicates the current locale.

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

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

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

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

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

MTEXTED
Sets the application for editing multiline text objects.

OLEQUALITY
Sets the default plot quality for OLE objects.

OLESTARTUP
Controls whether the source application of an embedded OLE object loads when plotting.
OSNAPCOORD
Controls whether coordinates entered on the command line will override running object snaps.

PAPERUPDATE
Controls the display of a warning dialog 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 at the Select Objects prompt.

PICKBOX
Sets the object selection target height, in pixels.

PICKDRAG
Controls the method of drawing a selection window.

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

PICKSTYLE
Controls the use of group selection and associative hatch selection.

PSTYLEPOLICY
Controls the plot style mode, Color-Dependent or Named, that is used when opening a drawing that was created in a release prior to AutoCAD 2000 or when creating a new drawing from scratch without using a drawing template.

QTEXTMODE
Controls how text is displayed.

RASTERPREVIEW
Controls whether BMP preview images are saved with the drawing.
ROLLOVERTIPS
Controls the display of rollover tooltips in the application.

SAVEFILEPATH
Stores the current automatic save file name.

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

SAVETIME
Sets the automatic save interval, in minutes.

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

TDUSRTIMER
Stores the user-elapsed timer.

TOOLTIPMERGE
Combines drafting tooltips into a single tooltip.

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

USERNAME
Specifies the user name.

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 display of toolbars and dockable windows, excluding the command window.

DRAGMODE
Controls the way dragged objects are displayed.

HIDEPALETTES
Hides all currently displayed palettes, including the command window.

LOGFILEOFF
Closes the text window log file opened by LOGFILEON.

LOGFILEON
Writes the text window contents to a file.

OPTIONS
Customizes the program settings.

REVDATE
Inserts or updates a block containing user name, current time and date, and drawing name.

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.

**EXEDIR**
Displays the folder path of the AutoCAD LT executable file.

**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.

LOCALE
Displays a code that indicates the current locale.

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

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

LOGFILEMODE
Specifies whether the contents of the text window are written to a log file.
LOGFILENAME
Specifies the path and name of the text window log file for the current drawing.

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

MTEXTED
Sets the application for editing multiline text objects.

OLEQUALITY
Sets the default plot quality for OLE objects.

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

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

PAPERUPDATE
Controls the display of a warning dialog 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 at the Select Objects prompt.

PICKBOX
Sets the object selection target height, in pixels.

PICKDRAG
Controls the method of drawing a selection window.
**PICKFIRST**
Controls whether you select objects before (noun-verb selection) or after you issue a command.

**PICKSTYLE**
Controls the use of group selection and associative hatch selection.

**PSTYLEPOLICY**
Controls the plot style mode, Color-Dependent or Named, that is used when opening a drawing that was created in a release prior to AutoCAD 2000 or when creating a new drawing from scratch without using a drawing template.

**QTEXTMODE**
Controls how text is displayed.

**RASTERPREVIEW**
Controls whether BMP preview images are saved with the drawing.

**ROLLOVERTIPS**
Controls the display of rollover tooltips in the application.

**SAVEFILEPATH**
Stores the current automatic save file name.

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

**SAVETIME**
Sets the automatic save interval, in minutes.

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

**TDUSRTIMER**
Stores the user-elapsed timer.

**TOOLTIPMERGE**
Combines drafting tooltips into a single tooltip.
TOOLTIPS
Controls the display of tooltips on the ribbon, toolbars, and other user interface elements.

USERNAME
Specifies the user name.

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.

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

- 2D Drafting & Annotation
- AutoCAD Classic
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 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 LT 2011\<release>\<product language>\Support\acadltSampleWorkspaces.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.
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
WSAUTOSAVE
- Saves changes you made to a workspace when you switch to another workspace.
WSCURRENT

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

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 `acadlt.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.

<table>
<thead>
<tr>
<th>Switch</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>/b</code></td>
<td>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.</td>
</tr>
<tr>
<td><code>/t</code></td>
<td>Template file name Creates a new drawing based on a template or prototype drawing. A DWT file type is assumed.</td>
</tr>
<tr>
<td><code>/c</code></td>
<td>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.</td>
</tr>
<tr>
<td><code>/v</code></td>
<td>View name Designates a particular view of the drawing for display at startup.</td>
</tr>
<tr>
<td><code>/w</code></td>
<td>Default workspace Designates which workspace in the loaded CUIx files should be restored on startup.</td>
</tr>
</tbody>
</table>

The syntax for using command line switches is

```
*drive:pathname\acadlt.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 LT 2011 with the drawing template arch1.dwt, restores a named view PLAN1, and executes a script file startup.scr.

```
d:\AutoCAD LT 2011\acadlt.exe" /t "d:\AutoCAD LT 2011\template\arch1" /v "plan1" /b "startup"
```

**NOTE** Command line switches override Options values 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 LT Properties dialog box, Shortcut tab, in the Target box, edit the parameters for the switch using the following syntax:

```
drive:pathname\acadlt.exe" ["drawing name"] [/switch "name"]
```

Valid switches are as follows:

<table>
<thead>
<tr>
<th>Switch</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>/b</td>
<td>Script name (b stands for batch process)</td>
</tr>
<tr>
<td>/t</td>
<td>Template file name</td>
</tr>
<tr>
<td>/c</td>
<td>Configuration folder</td>
</tr>
<tr>
<td>/v</td>
<td>View name</td>
</tr>
<tr>
<td>/w</td>
<td>Default workspace</td>
</tr>
</tbody>
</table>

For example, enter "d:\AutoCAD LT 2011\acadlt.exe" /t "d:\AutoCAD LT 2011\template\arch1" /v "plan1" /b "startup"

3. Click OK.

---

**Migrate and Specify Initial Setup**

Migrating from an older release or getting started with AutoCAD LT for the first time present different challenges.

Migrate Custom Settings allows you to copy your custom settings and files from a previous release of AutoCAD LT to the latest release, making it easier
to start using the new release. Initial Setup helps you to perform some basic customization of AutoCAD LT.

**Migrate Custom Settings and Files**

One of the challenges when moving from a previous release of AutoCAD LT to the latest release is getting the new release to look and behave like the previous release. You can use Migrate Custom Settings and the Customize User Interface (CUI) Editor to transfer many of your custom settings and files to the latest release.

**Initial Setup**

You can perform some basic customization of AutoCAD LT with Initial Setup which makes it easier to access specific tools and start a new drawing. Initial Setup is accessed from the Options dialog box. The following can be done with Initial Setup:

- Specify an industry that best describes your work to help search for related content to use in drawings and companion products developed by partners.
- Specify a drawing template to use when creating new drawings.

For more information on Initial Setup, see Customize AutoCAD LT with Initial Setup in the *Stand-Alone Installation Guide*.

**See also:**

- Migrate Custom Settings and Files from Previous Releases in the *Stand-Alone Installation Guide*
- Basic Customization in the *Customization Guide*
- User Interface Customization in the *Customization Guide*
Start and Save Drawings
Start a Drawing

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 151.

Choose a Drawing Template File

When you start a new drawing, AutoCAD LT 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.

1. **Choose a drawing template file**
   - Start a new drawing, choose a drawing template file similar to what you need for your discipline and units of measurement.

2. **Specify units and precision**
   - Specify a unit format and precision.

3. **Define various styles**
   - Specify values and settings for text, dimensions, and other styles.

4. **Specify additional settings**
   - Determine other settings, including layers, grid spacing, linetypes, and layouts.

5. **Save the drawing template file**
   - Save the file in the Template folder using a DWT file extension.

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 998

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 LT. 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 151
- Title blocks and borders on page 1079
- Layer names on page 313
- Snap and Grid spacing on page 420
- Text styles on page 901
- Dimension styles on page 961
- Multileader styles on page 886
- Table styles on page 942
- Linetypes on page 359
- Lineweights on page 370
- Layouts on page 265
- Page setups on page 1060

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 1342
To start a drawing by selecting a template file

1. Click the Application menu, and click New ➤ 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 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 92.
To display drawing properties for the active drawing

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

2. In the Drawing Properties dialog box, click tabs to view the different types of information.

To define drawing properties

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

2. In the Drawing Properties dialog box, enter property information on the following tabs:
   - **Summary tab.** Enter the drawing title, subject, author, keywords, comments, and a default address for hyperlinked data in your drawing. For example, you can add the keyword *Autodesk* to certain drawing files and then use DesignCenter to search for all drawing files with that keyword. For a hyperlink base, you can specify an Internet address or a path to a folder on a network drive.
   - **Custom tab.** Click Add. In the Add Custom Property dialog box, enter a name and a value for the custom property. Click OK. The new custom property and its value are displayed on the Custom tab. This information can be used for advanced searches in DesignCenter.

3. Click OK.

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. At the Command prompt, enter `geographiclocation`.
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. At the Command prompt, enter `geographiclocation`.
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 At the Command prompt, enter `geographiclocation`.
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 At the Command prompt, enter `geographiclocation`.
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

You can use several methods to find and open drawings, even damaged drawings. You can save and backup drawings automatically.

Open a Drawing

You open drawings to work on them just as you do with other Windows applications. In addition, you can choose from several alternative methods.

To open a drawing, you can

- Use Open on the File menu or Quick Access toolbar to display the Select File dialog box. If the FILEDIA system variable is set to 0, the Command prompt version displays instead of a file navigation dialog box.
- Double-click a drawing in Windows Explorer to launch AutoCAD LT® 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 LT. 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.
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.

<table>
<thead>
<tr>
<th>Missing Reference Types</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>External references</td>
<td>Missing external references are the result of AutoCAD LT not being able to resolve the last known location of an xref, raster image, or underlay. To resolve a missing external reference, locate the file and update its location using the External References palette. For information about resolving missing referenced drawing files, see Resolve Missing External References on page 1205. For information about working with raster images and underlays, see Attach Raster Image Files on page 1276 and Attach Files as Underlays on page 1244.</td>
</tr>
<tr>
<td>Shapes</td>
<td>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</td>
</tr>
</tbody>
</table>
Missing Reference Types  Description

<table>
<thead>
<tr>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>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 Line-types in the Customization Guide.</td>
</tr>
</tbody>
</table>

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 1356.

**Work with Large Objects**

AutoCAD LT 2010 supports object size limits greater than those available in previous releases. With increased object size limits you can create larger and more complex models. Using increased object size limits can result in compatibility issues with legacy drawing file formats (AutoCAD LT 2007 and earlier).

When working with drawings that you might need to exchange with others using AutoCAD LT 2009 and earlier, set the LARGEOBJECTSUPPORT system variable to 0. Setting LARGEOBJECTSUPPORT to 0 warns you when a drawing contains large objects that cannot be opened by a release of the program prior to AutoCAD LT 2010.

**TrustedDWG™ Drawing Files**

DWG, DWT, and DWS files created with Autodesk applications and RealDWG™-based applications are trusted by Autodesk. When you open a TrustedDWG file, the following icon displays in the application status bar or the drawing status bar.

If the DWGCHECK system variable is set to On (1), an alert box is displayed if

- The drawing file format is AutoCAD Release 14 or later and
- The drawing file was not originally created by an Autodesk application or RealDWG-based application.

For more information about TrustedDWG, click the TrustedDWG icon.

**Recover Defective Drawing Files**

In some circumstances, it is possible that a drawing file becomes defective. This can result from hardware problems or transmission errors. If a drawing
file is corrupt, you might be able to recover it. See Repair, Restore, or Recover Drawing Files on page 189.

**Change the Default Drawing Folder**

Each time you start AutoCAD LT, the My Documents folder is the default path in each standard file selection dialog box. Alternatively, you can configure AutoCAD LT to always default to a specified path by changing the default drawing folder using the REMEMBERFOLDERS system variable.

See also:

- Overview of DesignCenter on page 89
- Open and Save Drawing Files from the Internet on page 1343
- Overview of Using Markups for Design Review on page 1369

**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 LT 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**

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.

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.

**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 171.

- **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 175.

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

- **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.
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 175.

**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

QVDRAWING
  Displays open drawings and layouts in a drawing in preview images.
QVDRAWINGCLOSE
  Closes preview images of open drawings and layouts in a drawing.
SYSWINDOWS
  Arranges windows and icons when the application window is shared with external applications.
UPDATETHUMBSNOW
  Manually updates thumbnail previews in the Sheet Set Manager and the Quick View tool.
QVDRAWINGPIN
  Controls the default display state of preview images of drawings.
UPDATETHUMBNAIL
  Controls updating of the thumbnail previews in the Sheet Set Manager and Quick View.
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 UPDATETHUMBAIL 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**

**PLOT**
Plots a drawing to a plotter, printer, or file.

**PUBLISH**
Publishes drawings to DWF, DWFx, and PDF files, or to plotters.

**QVLAYOUT**
Displays preview images of model space and layouts in a drawing.

**QVLAYOUTCLOSE**
Closes preview images of model space and layouts in a drawing.

**UPDATETHUMBSNOW**
Manually updates thumbnail previews in the Sheet Set Manager and the Quick View tool.

**QVLAYOUTPIN**
Controls the default display state of preview images of model space and layouts in a drawing.

**UPDATETHUMBAIL**
Controls updating of the thumbnail previews in the Sheet Set Manager and Quick View.
Transfer Information between Open Drawings

You can easily transfer information between drawings that are open in a single session.

When you open multiple drawings in a single session, you can

■ Reference other drawings.
■ Copy and paste between drawings.
■ Drag selected objects from one drawing to another with the right-click button on your pointing device.
■ Use Match Properties (MATCHPROP) to copy properties from objects in one drawing to objects in another drawing.
■ Use object snaps, the Copy with Basepoint (COPYBASE) command, and the Paste to Original Coordinates (PASTEORIG) command to ensure accurate placement.

To switch between open drawings

Do one of the following to switch between open drawings:

■ On the status bar, click the Quick View Drawings tool.
■ Click anywhere in a drawing to make it active.
■ Use Ctrl+F6 or Ctrl+Tab.

Quick Reference

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 Microsoft Windows applications. You can also set up automatic saving and backup files and save only selected objects.

When you work on a drawing, you should save it frequently. Saving protects you from losing work in the event of a power failure or other unexpected event. If you want to create a new version of a drawing without affecting the original drawing, you can save it under another name.

The file extension for drawing files is .dwg, and unless you change the default file format in which drawings are saved, drawings are saved in the latest drawing file format. This format is optimized for file compression and for use on a network.

The character limit for a DWG file name (including its path) is 256 characters.

**NOTE** If the FILEDIA system variable is set to 0, the Command prompt version displays instead of a file navigation dialog box.

Save Part of a Drawing File

If you want to create a new drawing file from part of an existing drawing, you use the WBLOCK command. With the command, you can select objects or specify a block definition in your current drawing and save them to a new drawing file. You can also save a description with the new drawing.

Save to a Different Type of Drawing File

You can save a drawing to an earlier version of the drawing format (DWG) or drawing interchange format (DXF), or save a drawing as a template file. Choose the format from Files of Type in the Save Drawing As dialog box.

Save with Visual Fidelity for Annotative Objects

When working with annotative objects, this option allows you to maintain visual fidelity for these objects when they are viewed in AutoCAD LT 2007 and earlier releases. Visual fidelity is controlled by the SAVEFIDELITY system variable.

If you work primarily in model space, it is recommended that you turn off visual fidelity (set SAVEFIDELITY to 0). However, if you need to exchange drawings with other users, and layout fidelity is most important, then visual fidelity should be turned on (set SAVEFIDELITY to 1).
NOTE The SAVEFIDELITY system variable does not effect saving a drawing to the AutoCAD LT 2010 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 LT 2007 or earlier releases, and then open the drawing in AutoCAD LT 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 LT 2008 and later releases in AutoCAD LT 2007 and earlier releases.

When this option is not selected, a single model space representation is displayed on the Model tab. More annotation objects may be displayed on the Model tab depending on the ANNOALLVISIBLE setting. Also, more objects may be displayed in paper space viewports at different sizes than in AutoCAD LT 2008 and later releases.

**Reduce the Time Required to Save a Drawing File**

You can reduce the time required to save a drawing file if you specify incremental saves rather than full saves. An incremental save updates only those portions of the saved drawing file that you changed.

When you use incremental saves, drawing files will contain a percentage of potentially wasted space. This percentage increases after each incremental save until it reaches a specified maximum, at which time a full save is performed instead. You can set the incremental save percentage in the Open and Save tab of the Options dialog box or by setting the value of the system variable ISAVEPERCENT. If you set the value of ISAVEPERCENT to 0, all saves are full saves.

To reduce the size of drawing files, it is recommended that you perform a full save (with ISAVEPERCENT set to 0) before transmitting or archiving a drawing.

**Work Internationally**

If you share drawing files with companies in other countries and regions, the drawing file names might contain characters that are not used in other languages.
If a drawing file is created in a different language version of Windows, the following will occur:

- If support for the language is installed, the file name characters are visible in Windows Explorer.
- If support for the language is not installed, the file name characters appear as a series of boxes in Windows Explorer and a dialog box is displayed offering to install the language pack.

In either case, you will be able to open the drawing file beginning with AutoCAD 2007 or AutoCAD LT 2007 because these products are Unicode-compliant applications.

**NOTE** If you share drawing files with companies using earlier releases of the product, you can avoid file name issues for Asian languages and languages that use accented characters. In those circumstances, do not use high ASCII values, or values of 80 hexadecimal and above, when creating a file name.

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**Maintain Compatibility with Large Object Limits**

Drawings saved to a legacy drawing file format (AutoCAD LT 2007 or earlier) do not support objects greater than 256MB. With the AutoCAD LT 2010 drawing file format, these limitations have been removed allowing you to save objects that are greater in size.

When saving to a legacy drawing file format (AutoCAD LT 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 LT 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 LT, 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 LT.

**See also:**
- Save Drawings to Previous Drawing File Formats on page 1301
- Work with Drawings in Earlier Releases on page 1297
- Export Drawings to Other File Formats
- Create Drawing Files for Use as Blocks on page 671
- Add Identifying Information to Drawings on page 158
- Create and Restore Backup Files on page 192
- Share Drawing Files Internationally on page 1347
- Open and Save Drawing Files from the Internet on page 1343

**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.
In the Options dialog box, Open and Save tab, under File Save, select Maintain Visual Fidelity for Annotative Objects.

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

BLOCK
Creates a block definition from selected objects.

OPTIONS
Customizes the program settings.

QSAVE
Saves the current drawing using the file format specified in the Options dialog box.

QUIT
Exits the program.

SAVE
Saves the drawing under the current file name or a specified name.

SAVEAS
Saves a copy of the current drawing under a new file name.

WBLOCK
Writes objects or a block to a new drawing file.

DWGCHECK
Checks drawings for potential problems when opening them.
DWGNAME
Stores the name of the current drawing.

DWGPREFFIX
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 158
- Access Content with DesignCenter on page 92

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.
Select one or more files from the search results. Click OK.

In the Select File dialog box, click Open.

**Quick Reference**

**OPEN**

Opens an existing drawing file.

**Specify Search Paths and File Locations**

You can set the search path to drawing support files such as text fonts, drawings, linetypes, and hatch patterns. You also can specify the location of temporary files, which is important when working in a network environment.

The Files tab of the Options dialog box is where you set the search path that is used by the program to find drawing support files such as text fonts, drawings, linetypes, and hatch patterns. The MYDOCUMENTSPREFIX system variable stores the location of the My Documents folder for the current user.

Using the Files tab of the Options dialog box, you can also specify the location of temporary files. Temporary files are created on disk, and then deleted when you exit the program. The temporary folder is set to the location that Microsoft Windows uses. If you plan to run this program from a write-protected folder (for example, if you work on a network or open files on a CD), specify a different location for your temporary files.

The temporary folder that you specify must not be write-protected, and the drive containing the folder should have sufficient disk space for the temporary files. It is recommended that you manually delete the files from this folder on a regular basis to ensure sufficient space is provided for temporary files. If not enough space is available for temporary files, you may experience errors or instability in the program.

If you want to use a file that contains custom interface elements, specify it in the Customizations Files item on the Files tab of the Options dialog box. The default customization file is acadlt.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

OPTIONS
Customizes the program settings.

MYDOCUMENTSPREFIX
Stores the full path to the My Documents folder for the user currently logged
on.
Repair, Restore, or Recover Drawing Files

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 aclt.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. 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 Options dialog box, on the Open and Save tab, you can specify that backup files are created when you save drawings. If you do, each time you save a drawing, the previous version of your drawing is saved to a file with the same name and a .bak file extension. The backup file is located in the same folder as the drawing file.

You can revert to your backup version by renaming the .bak file in Windows Explorer to a file with a .dwg extension. You may want to copy it to a different folder to avoid overwriting your original file.

Save Your Drawing Automatically at Specified Intervals

If you turn the automatic save option on, your drawing is saved at specified time intervals. By default, files saved automatically are temporarily assigned the name filename_a_b_nnnn.sv$.

- **Filename** is the current drawing name.
- **a** is the number of open instances of the same drawing file in the same work session.
- **b** is the number of open instances of the same drawing in different work sessions.
- **nnnn** is a random number.

These temporary files are automatically deleted when a drawing closes normally. In the event of a program failure or a power failure, these files are not deleted.

To recover a previous version of your drawing from the automatically saved file, rename the file using a .dwg extension in place of the .sv$ extension before you close the program.

See also:

- Recover from a System Failure on page 194

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 LT. 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)

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.
Control the Drawing Views
Change Views

You can magnify the details in your drawing for a closer view or shift the view to a different part of the drawing. If you save views by name, you can restore them later.

Pan or Zoom a View

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

With the Realtime option of PAN, you pan dynamically by moving your pointing device. Like panning with a camera, PAN does not change the location or magnification of objects on your drawing; it changes only the view.

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

When you work with minute parts in your drawing, you may need to zoom out frequently to see an overview of your work. Use ZOOM Previous to return quickly to the prior view.

The options described here are the options most commonly used. For a description of all ZOOM options, see the Command Reference.

[Diagrams of zoomed out and zoomed in views]
**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.
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

**DSVIEWER**
- Opens the Aerial View window.

**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.

ZOOMFACTOR
Controls how much the magnification changes when the mouse wheel moves forward or backward.

Pan and Zoom with the Aerial View Window
In a large drawing, you can pan and zoom quickly in a window that can display the whole drawing.

You can use the Aerial View window to change the view in your current viewport quickly. If you keep the Aerial View window open as you work, you can zoom and pan without interrupting your current command. You can also specify a new view without having to choose a menu option or enter a command.

Use the View Box to Pan and Zoom
Within the Aerial View window is a view box, a heavy rectangle that displays the boundary of the view in your current viewport. You can change the view in the drawing by changing the view box in the Aerial View window. To zoom in to the drawing, make the view box smaller. To zoom out of the drawing, make the view box larger. All pan and zoom operations are performed by left-clicking. Right-click to end a pan or zoom operation.
To pan the drawing, move the view box.

**Change the View Displayed Inside the Aerial View Window**

You can use the Aerial View toolbar buttons to change the magnification of the image in the Aerial View window, or you can resize the image incrementally. These changes do not affect the view in the drawing itself.

**Use the Aerial View Window with Multiple Viewports**

Only the view in the current viewport is used in the Aerial View window.
The Aerial View image is updated as you make changes to your drawing and as you select different viewports. In complex drawings, you may want to turn off this dynamic updating to improve drawing speed. If you turn off this feature, the Aerial View image is updated only when you activate the Aerial View window.

To zoom to a new area using the Aerial View window
1. Click View menu ➤ Aerial View.
2. In the Aerial View window, click inside the view box until you see the arrow.
3. Drag to the right to zoom out. Drag to the left to zoom in.
4. Right-click to end the zoom operation.

To pan using the Aerial View window
1. Click View menu ➤ Aerial View.
2. In the Aerial View window, click inside the view box until you see an X.
3. Drag to change the view.
4. Right-click to end the pan operation.

To display the entire drawing in the Aerial View window
1. Click View menu ➤ Aerial View.
2. In the Aerial View window, click View menu ➤ Global.

To increase or decrease magnification of the Aerial View image
1. Click View menu ➤ Aerial View.
2. On the Aerial View toolbar, click Zoom Out or Zoom In.

**NOTE** When the entire drawing is displayed in the Aerial View window, the Zoom Out menu option and button are unavailable. When the current view nearly fills the Aerial View window, the Zoom In menu option and button are unavailable.

To turn dynamic updating on and off
1. Click View menu ➤ Aerial View.
In the Aerial View window, click Options menu ➤ Dynamic Update. A check mark indicates that the Aerial View window shows changes as they occur.

To turn viewport updating on and off
■ Click View menu ➤ Aerial View.
■ In the Aerial View window, click Options menu ➤ Auto Viewport. A check mark indicates that Aerial View displays the current viewport as you switch viewports.

Quick Reference

Commands
DSVIEWER
Opens the Aerial View window.

Pan and Zoom with SteeringWheel
SteeringWheel is a tracking menu where you to access different 2D navigation tools from a single tool.

Overview of SteeringWheel
The SteeringWheel, also known as a wheel, saves you time by combining 2D navigation tools into a single interface.

A wheel is divided into different sections known as wedges. Each wedge on a wheel represents a single navigation tool.

2D Navigation Wheel
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.

To display a wheel

Do one of the following:

■ Click View menu ➤ SteeringWheels.
■ Right-click over the drawing window and click SteeringWheels.
■ On the status bar, click SteeringWheels.

To close a wheel

Use one of the following methods to close the wheel:

■ Press the Esc or Enter
■ Click the Close button
■ Right-click the wheel and click Close Wheel

Quick Reference

Commands

NAVSWHEEL

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

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.

**Quick Reference**

**Commands**

NAVSWHEEL

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

**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.
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 wheel Pan tool

1. Display the 2D Navigation wheel.
2. Click the Pan wedge. Hold down the button on your pointing device and drag to reposition the model.
3. Release the button on your pointing device to return to the wheel.
4. Click Close to exit 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.

To restore the previous view

1. Display the 2D Navigation wheel.
2. Click the Rewind wedge.

To restore a previous view with the Rewind History panel

1. Display the 2D Navigation 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 that are 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.

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.

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 the 2D Navigation wheel.
2 Click the Zoom wedge.
3 Click Close to exit the wheel.

To zoom a view in and out by dragging
1 Display the 2D Navigation wheel.
2 Click and hold down the Zoom wedge.
The mouse pointer changes to the Zoom mouse pointer.
3 Drag the pointing device vertically to zoom in or out.
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.
Save and Restore Views

When you save specific views by name, you can restore them for layout and plotting or when you need to refer to specific details. You can create and save views using the VIEW command.

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

Named views are saved with a drawing and can be used any time. When you are composing a layout, you can restore a named view 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
■ The location of the view (the Model tab or a specific layout tab)
■ Layer visibility in the drawing at the time the view is saved
■ User coordinate system

Restore a Named View

You restore a named view to the current viewport. You can use named views to do the following:

■ Compose a layout.
■ Restore a view that you use frequently while you are working in model space.
■ Control which model space view is displayed when the drawing is opened.

To restore the previous view

■ Click View tab ➤ Navigate panel ➤ 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 restore a saved view

1. Do one of the following:
   - If you have more than one viewport in model space, click inside the viewport that contains the view you want to restore.
   - If you are working in a layout, select the viewport.

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

To rename a view

1. Click View tab ➤ Views panel ➤ Named Views.
2. In the View Manager, click the view name you want to change. If the view isn’t already displayed, expand the appropriate View list, and then click a view name.
3. In the General section of the Properties panel, select the view name. Enter a new name.
4. Click OK.

To change the properties of a view

1. Click View tab ➤ Views panel ➤ Named Views.
2. In the View Manager, click the view name you want to change. If the view isn’t already displayed, expand the appropriate View list, and then click a view name.
3. In the Properties panel, click the property that you want to change.
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 ➤ Named Views.
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.

**To restore the default view and UCS**

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

### Quick Reference

**Commands**

- VIEW
- VPORNS
  - 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.
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.

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 or PLAN to define paper space views. The view in paper space is always a plan view. Use the PERSPECTIVE system variable to turn off perspective views in drawings created in AutoCAD.

Quick Reference

Commands

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 VPOINT command 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 ➤ Named Views.
- Select a preset view (Top, Bottom, Left, and so on).

Quick Reference

Commands

VIEW

Saves and restores named 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.

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**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 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 VPOINT command 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; 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 VPOINT command is relative to the WCS (default) or the current UCS.

**Hide Lines or Shade 3D Objects**
You can create a hidden-line representation or a simple shaded picture of the objects displayed in the current viewport.

**Hide Lines in 3D Objects**
Suppress the display of objects—partly or entirely—that are located behind other objects in three-dimensional views.

You can create a hidden-line representation of the objects displayed in the current viewport. Hidden-line representations suppress lines, edges, and other objects—partly or entirely—that are located behind the following types of objects:

- Objects with nonzero thickness
- Circles
- Two-dimensional solids
- Wide polylines
- Surfaces and 3D solids (when viewing models created in AutoCAD)

The illustration below was created with lines that were extruded by giving them a nonzero thickness.
Objects on layers that are turned off but not frozen can also hide other objects. You can adjust how hidden lines are displayed by changing the settings in the Hidden Line Settings dialog box.

**Display Views with Hidden Lines Removed**

For viewing, you can temporarily suppress hidden lines with HIDE. HIDE suppresses all hidden lines in a view. When the view is regenerated, all objects are displayed normally.

See also:
- Set Options for Plotted Objects on page 1104

**To remove all hidden lines from a view**

1. Click View menu ➤ Hide.
2. The hidden lines reappear when the drawing is regenerated.

**To change the display properties of hidden lines**

1. At the Command prompt, enter `hlsettings`.
2. In the Hidden Line Settings dialog box, change the settings.
3. Click OK.

**To restore hidden lines in a view**

1. Click View menu ➤ Regen.
2. Hidden objects in the current viewport are restored to their normal display.

**To hide lines in layout views**

1. Select the viewport in which you want to apply hidden lines.
2 Click View tab ➤ Palettes panel ➤ Properties palette.
3 On the Properties palette, Misc area, Shade Plot list, select Hidden.

**To show hidden lines in the current viewport**
1 Select the viewport in which you want to show hidden lines.
2 Click View tab ➤ Palettes panel ➤ Properties palette.
3 On the Properties palette, Misc area, Shade Plot list, select As Displayed.

**To turn hidden lines on or off for printing objects not in layout views**
1 Click Output tab ➤ Plot panel ➤ Page Setup.
2 In the Page Setup Manager, select a page setup and click Modify.
3 In the Page Setup dialog box under Plot Options, select Hide Paperspace Objects.

**Quick Reference**

**Commands**

**HIDE**
Regenerates a 3D wireframe model with hidden lines suppressed.

**HLSETTINGS**
Changes the display properties of hidden lines.

**MVIEW**
Creates and controls layout viewports.

**PAGESETUP**
Controls the page layout, plotting device, paper size, and other settings for each new layout.

**REGEN**
Regenerates the entire drawing from the current viewport.
**System Variables**

**HALOGAP**
Specifies a gap to be displayed where an object is hidden by another object.

**HIDEPRECISION**
Controls the accuracy of hides and shades.

**HIDETEXT**
Specifies whether text objects created by the TEXT or MTEXT command are processed during a HIDE command.

**INTERSECTIONCOLOR**
Specifies the color of intersection polylines.

**INTERSECTIONDISPLAY**
Specifies the display of intersection polylines.

**OBSCURED COLOR**
Specifies the color of obscured lines.

**OBSCUREDLTYPE**
Specifies the linetype of obscured lines.

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**Add Simple Shading to 3D Objects**

Generate a simple shaded picture of the objects displayed in the current viewport.

You can create a simple shaded picture of the objects displayed in the current viewport. Shading fills certain objects with a solid color and removes the display of lines that are located *behind* the shaded objects. Each object is shaded using its current color.

In any view, you can shade the following types of objects:

- Objects with non-zero 3D thickness
- Circles
- Solid-filled polygons
- Wide polylines
- Surfaces and 3D solids (when viewing models created in AutoCAD)
Choose Shading Options

On systems that display fewer than 256 colors, SHADE produces an image that removes hidden lines and displays the faces in their original color with no lighting effect.

You have four shading options:

■ **256 Color.** Creates shaded faces with no edge highlighting. You need a 256-color display to see the full effect of this option.

■ **256 Color Edge Highlight.** Creates shaded faces with edges highlighted in the background color. You need a 256-color display with the program’s standard 256-color map to see the full effect of this option.

■ **16 Color Hidden Line.** Simulates a hidden-line display. The faces of polygons are painted in the background color, and the color of visible edges is determined by the color of the object. You can use any kind of monitor, including monochrome.

■ **16 Color Filled.** Draws faces in their original color but does not shade them. The program hides hidden lines and traces visible edges in the background color. You can use this setting on any kind of monitor.

**NOTE** The SHADE command produces only flat shading.

To create a shaded view

■ Click View menu ➤ Shade. Click a shading method.
  Objects in the current viewport are shaded with their current color.

To remove flat shading from a view

■ Click View menu ➤ Regen.
  Shaded objects in the current viewport are restored to their normal display.

Quick Reference

**Commands**

**HIDE**

Regenerates a 3D wireframe model with hidden lines suppressed.
REGEN
Regenerates the entire drawing from the current viewport.

SHADE
Displays a flat-shaded image of the drawing in the current viewport.

SHADEMODE
Controls the display of solid object shading in the current viewport.

System Variables

INTERSECTIONCOLOR
Controls the color of polylines at the intersection of 3D surfaces when the visual style is set to 2D Wireframe.

INTERSECTIONDISPLAY
Controls the display of polylines at the intersection of 3D surfaces when the visual style is set to 2D Wireframe.

SHADEEDGE
Controls the shading of edges.

SHADEF
Sets the ratio of diffuse reflective light to ambient light.
Use Viewing Tools

Navigate your drawing in different views with the viewing tools.

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.

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 ➤ 2D Wheel.

**To close a wheel**

Do one of the following:

- Press Esc or Enter
- Click the Close button

**Quick Reference**

**Commands**

NAVSWHEEL

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

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

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.
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 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:

■ **SteeringWheels.** Collection of wheels that offer rapid switching between specialized navigation tools.

■ **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.

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.
Reposition and Reorient the Navigation Bar

The navigation bar can be freely aligned along the edges of the current window. Drag the grip handle on the navigation bar to reposition it along one of the sides of the current 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 freely reposition the navigation bar along the edge of the current window

1. Click the grip handle displayed along the top of the navigation bar.
2. Drag the navigation bar along the edge of the window where you want it displayed.
3. 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

Command

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 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

Command

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.

You can change the behavior of the 3Dconnexion 3D mouse from the navigation bar.

### 3Dconnexion options on the navigation bar

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D Mode</td>
<td>Navigates the view using only 2D navigation options. The view moves in the direction of the controller cap.</td>
<td>Move the controller cap to pan and zoom the view.</td>
</tr>
<tr>
<td>3Dconnexion Settings</td>
<td>Displays 3Dconnexion Settings dialog box to change the settings of the 3Dconnexion 3D mouse.</td>
<td>Adjust the navigation speed used for view changes.</td>
</tr>
</tbody>
</table>

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

**Command**

`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

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

Set Model Space Viewports

On the Model tab, you can split the drawing area into one or more adjacent rectangular views known as model space viewports.

Viewports are areas that display different views of your model. As you work on the Model tab, you can split the drawing area into one or more adjacent rectangular views known as model space viewports. In large or complex drawings, displaying different views reduces the time needed to zoom or pan in a single view. Also, errors you might miss in one view may be apparent in the others.

Viewports created on the Model tab completely fill the drawing area and do not overlap. As you make changes in one viewport, the others are updated simultaneously. Three model space viewports are shown in the illustration.
You can also create viewports on a layout tab. You use those viewports, called
*layout viewports*, to arrange the views of your drawing on a sheet. You can move
and resize layout viewports. By using layout viewports, you have more control
over the display; for example, you can freeze certain layers in one layout
viewport without affecting the others. For more information about layouts
and layout viewports, see Create Multiple-View Drawing Layouts (Paper Space)
on page 263.

**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.
- Draw from one viewport to another when executing a command.
- Name a viewport arrangement so that you can reuse it on the Model tab
  or insert it on a layout tab.

Setting up different coordinate systems in individual viewports is useful if you
typically work on 3D models. See Assign User Coordinate System Orientations
to Viewports.

**Split and Join Model Space Viewports**

The illustrations below show several default model space viewport
configurations.
You can easily modify model space viewports by splitting and joining them. If you want to join two viewports, they must share a common edge of the same length.

**To subdivide a viewport on the Model tab**

1. If you have more than one viewport, click inside the viewport you want to subdivide.

2. To indicate how many model space viewports should be created, do one of the following:
   - Click View menu ➤ Viewports ➤ 2 Viewports.
   - Click View menu ➤ Viewports ➤ 3 Viewports.
   - Click View menu ➤ Viewports ➤ 4 Viewports.

3. At the Next prompt, specify the arrangement of the new viewports.

**To join two viewports on the Model tab**

1. Click View tab ➤ Viewports panel ➤ Viewports, Join.

2. Click within the model space viewport containing the view you want to keep.

3. Click within an adjacent viewport to join it to the first viewport.
To restore a single viewport on the Model tab

- Click View tab ➤ Viewports panel ➤ New.

To switch from a layout tab to the Model tab
- Click the Model tab at the bottom of the drawing area.

Quick Reference

Commands
MODEL
   Switches from a layout tab to the Model tab.
VPORTS
   Creates multiple viewports in model space or paper space.

System Variables
MAXACTVP
   Sets the maximum number of viewports that can be active at one time in a layout.
CTAB
   Returns the name of the current (model or layout) tab in the drawing.
TILEMODE
   Makes the Model tab or the last layout tab current.

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.

![Diagram of line drawn in two viewports]

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**

**CVPOR**
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 Tab Viewport Arrangements

Arrangements of model viewports can be saved and restored by name.

You don’t have to set up viewports and views every time you need them. With VPORTS, viewport arrangements can be saved and later restored by name. Settings that are saved with viewport arrangements include

- The number and position of viewports
- The views that the viewports contain
- The grid and snap settings for each viewport
- The UCS icon display setting for each viewport

You can list, restore, and delete the available viewport arrangements. A viewport arrangement saved on the Model tab can be inserted on a layout tab.

To save and name a viewport arrangement

1. Click View tab ➤ Viewports panel ➤ New.
2. In the Viewports dialog box, New Viewports tab, enter a name for the viewport configuration in the New Name box.
   The name can be up to 255 characters long and contain letters, digits, the special characters dollar sign ($), hyphen (-), and underscore (_).
3. Click OK.

NOTE You can save a viewport arrangement only on the Model tab.

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

RENAMEm
   Changes the names assigned to items such as layers and dimension styles.

VPORTSm
   Creates multiple viewports in model space or paper space.
Organize Drawings and Layouts
Create Single-View Drawings (Model Space)

If you are going to create a two-dimensional drawing that has one view, you can create the drawing and its annotation entirely in model space. This is the traditional method for creating drawings with AutoCAD LT®.

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 LT, there are two distinct working environments that are represented by the Model and layout tabs. These tabs are located near the bottom of the drawing area.

If you are going to create a two-dimensional drawing that has one view, you can create both the model and its annotation entirely in model space, not using a layout tab. This is the traditional method for creating drawings with AutoCAD LT. This method is simple but has several limitations, including

- It is suitable for 2D drawings only
- It does not support multiple views and view-dependent layer settings
- Scaling the annotation and title block requires computation unless you use objects.
With this method, you always draw geometric objects at full scale (1:1) and text, dimensions, and other annotation at a scale that will appear at the correct size when the drawing is plotted.

For information about using annotative objects and scaling annotations automatically, see Scale Annotations on page 764.

See also:

- Create Multiple-View Drawing Layouts (Paper Space) on page 263

Quick Reference

Commands

MODEL
Switches from a layout tab to the Model tab.

RENAME
Changes the names assigned to items such as layers and dimension styles.

VPORTS
Creates multiple viewports in model space or paper space.

System Variables

CVPORT
Displays the identification number of the current viewport.

MAXACTVP
Sets the maximum number of viewports that can be active at one time in a layout.

TILEMODE
Makes the Model tab or the last layout tab current.

VIEWCTR
Stores the center of view in the current viewport.

VIEWSIZE
Stores the height of the view displayed in the current viewport, measured in drawing units.
Draw, Scale, and Annotate in Model Space

If you draw and plot from model space, you must determine and apply a scale factor to annotate objects before you plot.

You can draw and plot entirely from model space. This method is useful primarily for two-dimensional drawings that have a single view. With this method, you use the following process:

■ Determine the unit of measurement (drawing units) for the drawing.
■ Specify the display style for the drawing unit.
■ Calculate and set the scale for dimensions, annotations, and blocks.
■ Draw at full scale (1:1) in model space.
■ Create the annotation and insert the blocks in model space.
■ Plot the drawing at the predetermined scale.

You can also use objects if you want to scale annotations automatically. For information about using annotative objects and scaling annotations automatically, see Scale Annotations on page 764.

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 dialog box (HATCH) or with the HPSCALE system variable.
- **Blocks.** Specify the insertion scale for blocks either as you insert them, or set an insertion scale in the Insert dialog box (INSERT) or in DesignCenter (ADCENTER). The system variables used for inserting blocks are INSUNITS, INSUNITSDEFSOURCE, and INSUNITSDEFTARGET. This also applies to the border and title block of the drawing.

You can also use objects if you want to scale annotations automatically. For information about using annotative objects and scaling annotations automatically, see Scale Annotations on page 764.

Determine the Scale Factor for Plotting

To plot your drawing from the Model tab, you calculate the exact scale factor by converting the drawing scale to a ratio of 1:n. This ratio compares plotted units to drawing units that represent the actual size of the objects you are drawing.

For example, if you plan to plot at a scale of 1/4 inch = 1 foot, you would calculate the scale factor 48 as follows:

\[
\frac{1/4''}{12''} = \frac{1}{12} \\
12 x \frac{1}{4} = 3 \\
1 \text{ (plotted unit)} = 48 \text{ (drawing units)}
\]

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.

<table>
<thead>
<tr>
<th>Scale</th>
<th>Scale factor</th>
<th>To plot text size at</th>
<th>Set drawing text size to</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 cm = 1 m</td>
<td>100</td>
<td>3 mm</td>
<td>30 cm</td>
</tr>
<tr>
<td>1/8' = 1'-0&quot;</td>
<td>96</td>
<td>1/8&quot;</td>
<td>12&quot;</td>
</tr>
<tr>
<td>3/16&quot; = 1'-0&quot;</td>
<td>64</td>
<td>1/8&quot;</td>
<td>8&quot;</td>
</tr>
<tr>
<td>1/4&quot; = 1'-0&quot;</td>
<td>48</td>
<td>1/8&quot;</td>
<td>6&quot;</td>
</tr>
<tr>
<td>3/8&quot; = 1'-0&quot;</td>
<td>32</td>
<td>1/8&quot;</td>
<td>4&quot;</td>
</tr>
<tr>
<td>1/2&quot; = 1'-0&quot;</td>
<td>24</td>
<td>1/8&quot;</td>
<td>3&quot;</td>
</tr>
<tr>
<td>3/4&quot; = 1'-0&quot;</td>
<td>16</td>
<td>1/8&quot;</td>
<td>2&quot;</td>
</tr>
<tr>
<td>1&quot; = 1'-0&quot;</td>
<td>12</td>
<td>1/8&quot;</td>
<td>1.5&quot;</td>
</tr>
<tr>
<td>1 1/2&quot; = 1'-0&quot;</td>
<td>8</td>
<td>1/8&quot;</td>
<td>1.0&quot;</td>
</tr>
</tbody>
</table>

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 151

**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.
DIMENT
   Creates and modifies dimension styles.
INSERT
   Inserts a block or drawing into the current drawing.
LINETYPE
   Loads, sets, and modifies linetypes.
PLOT
   Plots a drawing to a plotter, printer, or file.
STYLE
   Creates, modifies, or specifies text styles.
UNITS
Controls coordinate and angle display formats and precision.

**System Variables**

CELTSCALE
Sets the current object linetype scaling factor.

DIMSCALE
Sets the overall scale factor applied to dimensioning variables that specify sizes, distances, or offsets.

HPSCALE
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

LUNITS
Sets linear units.

TEXTSIZE
Sets the default height for new text objects drawn with the current text style.
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. These are represented by the Model and layout tabs.

Typically, a model composed of geometric objects is created in a three-dimensional space called *model space*. A final layout of specific views and annotations of this model is created in a two-dimensional space called *paper space*. These spaces are accessible on two or more tabs near the bottom of the drawing area: the Model tab and one or more layout tabs.

*NOTE* These tabs can be hidden, appearing instead as buttons on the status bar at the bottom-center of the application window.

Working on the Model tab, you draw a model of your subject at 1:1 scale. Working on a layout tab, you can create one or more *layout viewports*, dimensions, notes, and a title block to represent a drawing sheet.
Each layout viewport is like a picture frame containing a “photograph” of the model in model space. Each layout viewport contains a view that displays the model at the scale and orientation that you specify. You can also specify which layers are visible in each layout viewport.

After you finish arranging the layout, you turn off the layer that contains the layout viewport objects. The views are still visible, and you can plot the layout without displaying the viewport boundaries.

**Quick Reference**

LAYOUT

- Creates and modifies drawing layout tabs.

LAYOUTWIZARD

- Creates a new layout tab and specifies page and plot settings.

MODEL

- Switches from a layout tab to the Model tab.

MSPACE

- 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 layout tab to prepare your drawing for plotting, you follow a series of steps in a process.

You design the subject of your drawing on the Model tab (in model space) and prepare it for plotting on a layout tab (in paper space).

There is one Model tab and one or more layout tabs at the bottom of the drawing window.

**NOTE** These tabs can be hidden, appearing instead as buttons on the status bar at the bottom-center of the application window.

You can initialize a layout by clicking on its tab to activate the previously unused layout. A layout does not contain any plot settings before initialization. Once initialized, layouts can be drawn upon, published, and added to sheet sets as sheets (after the drawing has been saved).
Process Summary
When you prepare a layout, you typically step through the following process:

- Create a model of your subject on the Model tab.
- Click a layout tab.
- Specify layout page settings such as plotting device, paper size, plot area, plot scale, and drawing orientation.
- Insert a title block into the layout (unless you have started with a drawing template that already has a title block).
- Create a new layer to be used for layout viewports.
- Create layout viewports and position them on the layout.
- Set the orientation, scale, and layer visibility of the view in each layout viewport.
- Add dimensions and annotate in the layout as needed.
- Turn off the layer containing the layout viewports.
- Plot your layout.

You can also use objects if you want to annotate your drawing in model space and scale the annotations automatically. For information about using annotative objects and scaling annotations automatically, see Scale Annotations on page 764.

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 layout tabs.

LAYOUTWIZARD
- Creates a new layout tab and specifies page and plot settings.

MODEL
- Switches from a layout tab to the Model tab.
MSPACE
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.

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 on the Model Tab

The Model tab accesses a limitless drawing area called model space. In model space, you draw, view, and edit your model.

In model space, you draw your model at 1:1 scale, and you decide whether one unit represents one millimeter, one centimeter, one inch, one foot, or whatever unit is most convenient or customary in your business.

On the Model tab, you can view and edit model space objects. The crosshairs cursor is active over the entire drawing area.

In model space, you can also define named views that you display in layout viewports on a layout.

To activate the Model tab

Do one of the following to make the Model tab current:

- Click the Model tab.
- Right-click any layout tab or the Model tab. Click Activate Model Tab.
- If the Model and layout tabs are hidden, click the Model button on the status bar at the bottom of the application window.
Quick Reference

MODEL
Switches from a layout tab to the Model tab.

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.

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 Layout Tab

Layout tabs access an area called paper space. In paper space, you place your title block, create layout viewports to display views, dimension your drawing, and add notes.

In paper space, one unit represents the paper distance on a plotted sheet. The units will be in either millimeters or inches, depending on the plot setup for your plotter.

On a layout tab, you can view and edit paper space objects, such as layout viewports and title blocks. You can also move an object (such as a leader or a title block) from model space to paper space (or vice versa). The crosshairs cursor is active over the entire layout area.
NOTE These tabs can be hidden, appearing instead as buttons on the status bar at the bottom-center of the application window. To display the tabs, right-click the Model or layout button and click Display Layout and Model Tabs on the shortcut menu.

Create Additional Layout Tabs

By default, a new drawing starts with two layout tabs, named Layout1 and Layout2. If you use a drawing template or open an existing drawing, the layout tabs in your drawing may be named differently.

You can create a new layout tab using one of the following methods:

■ Add a new layout tab with no settings and then specify the settings in the Page Setup Manager.
■ Use the Create Layout wizard to create the layout tab and specify settings.
■ Copy a layout tab and its settings from the current drawing file.
■ Import a layout tab from an existing drawing template (DWT) file or drawing (DWG) file.

NOTE You can create multiple layouts in a drawing; each layout can contain different plot settings and paper sizes. However, to avoid confusion when transmitting and publishing drawings, it is usually recommended that you create only one layout for each drawing.

Use the Layout Wizard to Specify Layout Settings

You can create a new layout using the Create Layout wizard. The wizard prompts you for information about the layout settings, including

■ A name for the new layout
■ The printer associated with the layout
■ A paper size to use for the layout
■ The orientation of the drawing on the paper
■ A title block
■ Viewport setup information
■ A location for the viewport configuration in the layout
You can edit the information entered in the wizard later. Click Output tab ➤ Plot panel ➤ Page Setup Manager.

**To 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 layout tabs.

**LAYOUTWIZARD**

Creates a new layout tab and specifies page and plot settings.

**MODEL**

Switches from a layout tab to the Model tab.

**MSPACE**

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 (model or layout) tab in the drawing.

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.

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 button on the status bar to maximize the layout viewport. The maximized layout viewport expands to fill the drawing area. The center point and the layer visibility settings of the viewport are retained, and the surrounding objects are displayed.

You can pan and zoom while you are working in model space, but when you restore the viewport to return to paper space, the position and scale of the objects in the layout viewport are restored.

**NOTE** If you use PLOT while a viewport is maximized, the layout tab is restored before the Plot dialog box is displayed. If you save and close the drawing while a viewport is maximized, the drawing opens with the layout tab restored.

If you choose to switch to the Model tab to make changes, the layer visibility settings are the settings for the drawing as a whole, not the settings for that particular layout viewport. Also, the view is not centered or magnified the same way it is in the layout viewport.

**Adjust the View in a Layout Viewport**

If you plan to pan the view and change the visibility of layers, double-click within a layout viewport to access model space. The viewport border becomes thicker, and the crosshairs cursor is visible in the current viewport only. All active viewports in the layout remain visible while you work. You can freeze and thaw layers in the current viewport in the Layer Properties Manager, and you can pan the view. To return to paper space, double-click an empty area on the layout outside a viewport. The changes you made are displayed in the viewport.

If you set the scale in the layout viewport before you access model space, you can lock the scale to prevent changes. When the scale is locked, you cannot use ZOOM while you work in model space.
To switch between model space and paper space on a layout

On a layout, use one of the following methods:

- If you are in paper space, double-click within a layout viewport.
  You are now in model space. The selected layout viewport becomes the current viewport, and you can pan the view and change layer properties. If you need to make significant changes to the model, it is recommended that you use VPMAX to maximize the layout viewport or switch to the Model tab.

- If you are in model space in a layout viewport, double-click outside the viewport.
  You are now in paper space. You can create and modify objects on the layout.

- If you are in model space and want to switch to another layout viewport, double-click within another layout viewport, or press Ctrl+R to cycle through the existing layout viewports.

To edit in a maximized layout viewport

1. Click the boundary of the layout viewport to select it.

   NOTE You can maximize a locked viewport and modify objects. When you restore the viewport, it is locked again.

2. On the status bar, click the Maximize Viewport button.
   You can restore the viewport and maximize another viewport by clicking one of the arrows next to the Maximize Viewport button.

3. Make any changes.

4. To return to the layout viewport, click the Restore Viewport button on the status bar.
   The center point and magnification are returned to the settings that were in effect before the viewport was maximized.

Quick Reference

MODEL

Switches from a layout tab to the Model tab.
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.

<table>
<thead>
<tr>
<th>Object Type</th>
<th>Representation in Exported Drawing</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dimensions</td>
<td>Dimensions that exceed the boundaries of the layout viewport are exploded.</td>
</tr>
<tr>
<td>Object Type</td>
<td>Representation in Exported Drawing</td>
</tr>
<tr>
<td>-------------------------------------------------</td>
<td>--------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Constraints</td>
<td>Dimensional constraints are removed.</td>
</tr>
<tr>
<td>Standard or dynamic block (with or without</td>
<td>Standard or dynamic blocks, with or without attributes, that exceed the boundaries of the layout</td>
</tr>
<tr>
<td>attributes)</td>
<td>viewport will be converted to a new, anonymous block. Attributes are converted to text objects in</td>
</tr>
<tr>
<td></td>
<td>the block.</td>
</tr>
<tr>
<td></td>
<td><strong>NOTE</strong> The “Allow Exploding” setting (a setting on the block definition) is ignored if the block</td>
</tr>
<tr>
<td></td>
<td>exceeds the boundaries of the layout viewport.</td>
</tr>
<tr>
<td>Annotative objects</td>
<td>Objects become non-annotative.</td>
</tr>
<tr>
<td>External reference (xref)</td>
<td>An Xref with nested objects that exceed the boundaries of the layout viewport is converted to a</td>
</tr>
<tr>
<td></td>
<td>block reference and exploded.</td>
</tr>
<tr>
<td>Layout viewport</td>
<td>Layout viewports are represented by either a polyline or the clipped viewport object.</td>
</tr>
<tr>
<td>Custom objects</td>
<td>Custom objects are exploded and converted to anonymous blocks.</td>
</tr>
<tr>
<td>Xref clipped with the XCLIP command</td>
<td>Xrefs clipped with the XCLIP command are converted to a clipped block reference.</td>
</tr>
<tr>
<td>Visual Styles</td>
<td>The 2D wireframe visual style is used.</td>
</tr>
<tr>
<td>Perspective viewports</td>
<td>Objects within a perspective viewport will have parallel projection.</td>
</tr>
</tbody>
</table>

**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 `exportlayout`.

**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 plot, you can turn off the layer and plot the layout without plotting the boundaries of the layout viewports.

With MVIEW, you have several options for creating one or more layout viewports. You can also use COPY and ARRAY to create multiple layout viewports.
Create Nonrectangular Layout Viewports

You can create a new viewport with nonrectangular boundaries by converting an object drawn in paper space into a layout viewport.

You can use the MVIEW command to create nonrectangular viewports.

- With the Object option, you can select a closed object, such as a circle or closed polyline created in paper space, to convert into a layout viewport. The object that defines the viewport boundary is associated with the viewport after the viewport is created.

- With the Polygonal option, you can create a nonrectangular layout viewport by specifying points. The prompts are the same as the prompts for creating a polyline.

**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.

Create and Modify Layout Viewports | 279
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:
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.

**Add Date and Time Stamps to Layouts**

You can track drawing revisions by inserting information that identifies the file name, time and date of the revision, and the name of the person who revised it.

The revision information is inserted in the form of attributes that have been saved in a block named `REVDATE`, and it is automatically updated as needed. The attributes contain the information; the block is a named grouping of the attributes. The attributes are

- `REVDATE`, the date and time
- `FNAME`, the drawing file name, not including the path and extension
- `USER`, the name of person revising the drawing

If you don’t insert the date and time stamp when you create the drawing, you can insert it later. A block is inserted at the point you specify. If necessary,
you can explode this block and reposition the individual attributes. By then saving the newly positioned attributes as a block named REVDATE, you can ensure that they will be updated whenever you use REVDATE.

**NOTE** You can also use the Block Editor to modify the REVDATE block.

By default, the user name used by the date and time stamp is the name specified during installation. If a different person works on the drawing, you can change the name used by REVDATE.

You can also change the time and date formats by choosing the Regional Settings icon in the Windows Control Panel folder.

**See also:**
- “Work with Blocks”

**To add a date and time stamp to an existing drawing**

1. At the Command prompt, enter `redate`.
   
   If a date and time stamp has already been inserted, it is updated. Otherwise, go to step 2.

2. Specify an insertion point for the date and time stamp.

3. Specify the angle of rotation (0 or 90 degrees) for the stamp.

**To change the user name displayed by the date and time stamp**

1. Click Tools menu ➤ Options.

2. On the System tab, change the entry listed under User Name.

**Quick Reference**

**REVDATE**

Inserts or updates a block containing user name, current time and date, and drawing name.

**TIME**

Displays the date and time statistics of a drawing.

**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.

Control Views in Layout Viewports
When you create a layout, you can add layout viewports that act as windows into model space. In each layout viewport, you can control the view that is displayed.

Scale Views in Layout Viewports
To scale each displayed view in the plotted drawing accurately, set the scale of each view relative to paper space.

You can change the view scale of the viewport using

- The Properties palette
- The XP option of the ZOOM command
- The Viewports toolbar

NOTE You can modify the list of scales that are displayed in all view and plot scale lists with SCALELISTEDIT.

When you work in a layout, the scale factor of a view in a layout viewport represents a ratio between the actual size of the model displayed in the viewport and the size of the layout. The ratio is determined by dividing the paper space units by the model space units. For example, for a quarter-scale...
drawing, the ratio would be a scale factor of one paper space unit to four model space units, or 1:4.

Scaling or stretching the layout viewport border does not change the scale of the view within the viewport.

When creating a new drawing based on a template, the scales in the template are used in the new drawing. The scales in the registry 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, PLAN, and VIEW, no longer function in that viewport.

**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 764.

**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.

Scale Views in Layout Viewports | 285
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 next to Display Locked, select 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.
Freeze Specified Layers in a Layout Viewport

A major benefit to using layout viewports is that you can selectively freeze layers in each layout viewport. You can also specify default visibility settings for new viewports and for new layers. As a result, you can view different objects in each layout viewport.

You can freeze or thaw layers in current and future layout viewports without affecting other viewports. Frozen layers are invisible. They are not regenerated or plotted. In the illustration, the layer showing terrain has been frozen in one viewport.

Thawing the layer restores visibility. The easiest way to freeze or thaw layers in the current viewport is to use the Layer Properties Manager.

In the Layer Properties Manager, on the right side, use the column labeled VP Freeze to freeze one or more layers in the current layout viewport. To display the VP Freeze column, you must be on a layout tab. Specify the current layout viewport by double-clicking anywhere within its borders.

Freeze or Thaw Layers Automatically in New Layout Viewports

You can set visibility defaults for specific layers in all new layout viewports. For example, you can restrict the display of dimensions by freezing the DIMENSIONS layer in all new viewports. If you create a viewport that requires dimensions, you can override the default setting by changing the setting in the current viewport. Changing the default for new viewports does not affect existing viewports.

Create New Layers That Are Frozen in All Layout Viewports

You can create new layers that are frozen in all existing and new layout viewports. Then you can thaw the layers in the viewports you specify. This is a shortcut for creating a new layer that is visible only in a single viewport.
To freeze or thaw layers in the current layout viewport
1 Double-click within a layout viewport to make it current.
2 Click Home tab ➤ Layer panel ➤ Layer Properties.
3 In the Layer Properties Manager, select the layers to freeze or thaw.
   Hold down Ctrl to select more than one layer. Hold down Shift to select
   a sequence of layers.
4 Click the icon in the VP Freeze column for one of the selected layers.
5 Click OK.

To view a list of layers that are frozen in the current viewport
1 Click a layout tab.
2 Double-click within a layout viewport to make it current.
3 Click Home tab ➤ Layer panel ➤ Layer Properties.
4 In the Layer Properties Manager, look at the VP Freeze column for the
   Freeze/Thaw in Current Viewports icon.
5 Click OK.

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.

6 Click OK.

**To freeze or thaw layers in all new viewports**

1 Click a layout tab.

2 Click Home tab ➤ Layer panel ➤ Layer Properties.

3 In the Layer Properties Manager, select the layer or layers you want to be automatically frozen or thawed in new viewports you create.

   Hold down Ctrl to select more than one layer. Hold down Shift to select a sequence of layers.

4 In the New VP Freeze column, click the icon to change a layer's state. The sun icon means that a layer is thawed; the snowflake icon means that a layer is frozen.

5 Click OK.

**To create new layers that are frozen in all viewports**

1 Click a layout tab.

2 Click Home tab ➤ Layer panel ➤ Layer Properties.

3 Click the New Layer button to create a layer.

4 Rename the new layer.

5 Click the icon in the Freeze column to change the layer's state to frozen. The sun icon means that a layer is thawed; the snowflake icon means that a layer is frozen.

6 Click OK.

**Quick Reference**

**LAYER**

Manages layers and layer properties.
Screen Objects in Layout Viewports

Screening refers to applying less ink to an object when it is plotted. The object appears dimmer on the screen and on the plotted paper. Screening can be used to help differentiate objects in a drawing without changing the objects' color properties.

To assign a screening value to an object, you must assign a plot style to the object, and then define the screening value in that plot style.

You can assign a screening value from 0 to 100. The default setting, 100, means no screening is applied, and the object is displayed with normal ink intensity. A screening value of 0 means the object contains no ink and is thus invisible in that viewport.

See also:

- Set Options for Plotted Objects on page 1104

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.
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.
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 the PSLTSCALE system variable to maintain the same linetype scaling for objects displayed at different zoom factors in a layout and in a layout viewport. For example, with PSLTSCALE set to 1 (default), set the current linetype to dashed, and then draw a line in a paper space layout. In the layout, create a viewport with a zoom factor of 1x, make that layout viewport current, and then draw a line using the same dashed linetype. The dashed lines should appear to be the same. If you change the viewport zoom factor to 2x, the linetype scaling for the dashed line in the layout and the dashed line in the layout viewport will be the same, regardless of the difference in the zoom factor.

With PSLTSCALE turned on, you can still control the dash lengths with LTSCALE and CELTSCALE. In the following illustration, the pattern of the linetypes in the drawing on the left has been scaled to be the same regardless of the scale of the view. In the drawing on the right, the scale of the linetypes matches the scale of each view.

See also:
■ Set the Lineweight Scale for a Layout on page 1072

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.

To adjust the views on a layout with precision, you can create construction geometry, use object snaps on the model-space objects displayed in layout.
viewports, or use one of the cursor constraint features available on the status bar.

**To align objects between viewports using a construction line**

1. Make sure you are on a layout tab.

2. Click Home tab ➤ Draw panel ➤ Construction Line.

3. Specify a point in the first viewport. Specify a second point to determine a line for the alignment.
   Choose a point that can be aligned with objects in the second viewport. Use object snaps for precision.

4. Click Home tab ➤ Modify panel ➤ Move.

5. Select the viewport to align to the first viewport. Press Enter.

6. When prompted for a base point, specify a point in the second viewport. Choose a point that corresponds with the point selected in the first viewport.

7. When prompted for the second point, hold down Shift and right-click. Click Object Snap menu ➤ Perpendicular.

8. Click on 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.

**Quick Reference**

MOVE
Moves objects a specified distance in a specified direction.

UCS
Manages user coordinate systems.
UCSICON
Controls the visibility and placement of the UCS icon.

UCSMAN
Manages defined user coordinate systems.

UCSICON
Controls the visibility and placement of the UCS icon.

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.

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
➤ Select the viewport with the view you want to rotate.

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).
4. Click View tab ➤ Coordinates panel ➤ World.
5. Click View tab ➤ Coordinates panel ➤ Z.
6. To rotate the view 90 degrees clockwise, enter 90. To rotate the view 90 degrees counterclockwise, enter -90.
7. 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.

Quick Reference

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

UCS
Manages user coordinate systems.

UCSICON
Controls the visibility and placement of the UCS icon.

UCSMAN
Manages defined user coordinate systems.

UCSICON
Controls the visibility and placement of the UCS icon.

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 Options dialog box, Support tab. The layout template has a .dwt or .dwg extension like a drawing template or drawing file, but it contains little information not essential to the layout.

When you create a new layout template, any named items, such as blocks, layers, and dimension styles, that are used in the layout are saved with the template. These definition table items are imported as part of the layout settings if you import this template into a new layout. It is recommended that you use the Save As option of the LAYOUT command to create a new layout template. When you use the Save As option, unused definition table items are not saved with the file; they are not added to the new layout into which you import the template.

If you insert a layout from a drawing or template that was not created using the Save As option of the LAYOUT command, definition table items that are
used in the drawing but not in the layout are inserted with the layout. To eliminate unnecessary definition table items, use the PURGE command.

**Insert a Layout Using DesignCenter**

Using DesignCenter™, you can drag a layout with its objects from any drawing into the current drawing.

When you use DesignCenter to insert a layout into a drawing, a new layout is created that includes all of the paper space objects, definition tables, and block definitions from the source layout. You can delete unneeded paper space objects. To eliminate any unnecessary definition table information from the new layout, use the PURGE command.

**To create a layout using a layout template**

1. Click Insert menu ➤ Layout ➤ Layout From Template.
2. In the Select Template From File dialog box, select a drawing template file from the list.
3. Click Open.
4. In the Insert Layout(s) dialog box, select the layout template from the list. Click OK.

A new layout is created using the layout template you selected. The new layout is assigned the name Layout with the next number in the sequence and with the name of the imported layout attached.

For example, if you insert a layout called ANSI D from a layout template and you already have two layouts in your drawing called Layout1 and Layout2, the new layout is called Layout3 - ANSI D.

**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 layout tabs.

PAGESETUP
Controls the page layout, plotting device, paper size, and other settings for each new layout.

PCINWIZARD
Displays a wizard to import PCP and PC2 configuration file plot settings into the Model tab or current layout.

PURGE
Removes unused items, such as block definitions and layers, from the drawing.

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.
Create and Modify Objects
Control the Properties of Objects

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.
See also:
- Control the Color and Linetype Properties in Blocks on page 674

**Quick Reference**

CHPROP
- Changes the properties of an object.

PROPERTIES
- Controls properties of existing objects.

PROPERTIESCLOSE
- Closes the Properties palette.

CECOLOR
- Sets the color of new objects.

CELTYPE
- Sets the linetype of new objects.

CELWEIGHT
- Sets the lineweight of new objects.

CETRANSPLANCENCY
- 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.

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 Quick Properties Palette

The Quick Properties palette lists the most commonly used properties for each object type or a set of objects. You can easily customize the quick properties for any object in the Customize User Interface (CUI) editor. See Quick Properties in the Customization Guide.

- When one or more objects of the same type are selected, the Quick Properties palette displays the selected properties of that object type.
- When two or more objects of different types are selected, the Quick Properties palette displays the common properties, if any for all objects in the selection set.

Use the Properties Palette

The Properties palette lists the current settings for properties of the selected object or set of objects. You can modify any property that can be changed by specifying a new value.

- When more than one object is selected, the Properties palette displays only those properties common to all objects in the selection set.
- When no objects are selected, the Properties palette displays only the general properties of the current layer, the name of the plot style table attached to the layer, the view properties, and information about the UCS.

You can double-click most objects to open the Properties palette when the DBLCLKEDIT system variable is set to On (the default). The exceptions include blocks and attributes, text, and xrefs. If you double-click any of these objects, an object-specific dialog box is displayed instead of the Properties palette.

**NOTE** The DBLCLKEDIT system variable must be set to On and the PICKFIRST system variable must be set to 1 (the default) for double-clicking to work.

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.

**IMPORTANT** For performance reasons, the number of objects controlled by the ribbon when changing an object property or displaying a contextual tab is limited to 2500 objects. You can change this limit by changing the value of the RIBBONCONTEXTSELLIM system variable.

**Change Object Property or ByBlock Settings to ByLayer**

Objects that have a ByBlock setting can also be changed to ByLayer. When an object’s properties are not set to ByLayer, those objects do not display the layer property overrides that were set by viewport.

See also:
- Control the Color and Linetype Properties in Blocks on page 674
- Select Objects by Properties on page 529
- Set Interface Options on page 107
- Quick Properties in the *Customization Guide*

**To change the settings of a Quick Properties palette**

1. Click Tools ➤ 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.

**SETBYLAYER**
Changes the property overrides of selected objects to ByLayer.

**CETRANSAPRENCY**
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.

QPLOCATION
Sets the location mode of Quick Properties palette.

QPMODE
Sets the on or off state of Quick Properties palette.

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, linewidth, 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 ➤ Properties panel ➤ Match Properties.
2. Select the object whose properties you want to copy.
3. If you want to control which properties are transferred, enter s (Settings). In the Property Settings dialog box, clear the items that you do not want copied (all are on by default). Click OK.
4. Select the objects to which you want to apply the selected properties and press Enter.

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.

**Quick Reference**

CLASSICLAYER

Opens the legacy Layer Properties Manager.
LAYER

Manages layers and layer properties.

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.

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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 transparency. 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 Color control on the Properties toolbar is set to BYLAYER, the color of new objects is determined by the color setting for the layer in the Layer Properties Manager.

If you set a specific color in the Color control, that color is used for all new objects, overriding the default color for the current layer. The same is true for the Linetype, Lineweight, and Plot Style controls on the Properties toolbar.

The BYBLOCK setting should be used only for creating blocks. See Control the Color and Linetype Properties in Blocks on page 674.

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 325 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 visibility 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 transparency 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.

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.
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, you can use wild-card characters in layer name filters when you need to find those layers quickly.

**NOTE** If you consistently use a specific layering scheme, you can set up a drawing template with layers, linetypes, and colors already assigned. For more information about creating templates, see Use a Drawing Template File on page 155.

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 Tools tab ➤ Drawing Utilities panel ➤ Purge.
   The Purge dialog box displays a tree view of object types with items that can be purged (removed from the drawing).
2. To purge unreferenced layers, use one of the following methods:
   ■ To purge all unreferenced layers, select Layers.
   ■ To purge specific layers, double-click Layers to expand the tree view. Select the layers to be purged.
3. If the item you want to purge is not listed, select View Items You Cannot Purge, select the layer, and read the explanation.
4. You are prompted to confirm each item in the list. If you do not want to confirm each purge, clear the Confirm Each Item to Be Purged option.
5. Click Purge.
6. To confirm the purging of each item, respond to the prompt by choosing Yes or No, or Yes to All if more than one item is selected.
7. Click Close.

Quick Reference

CLASSICLAYER
Open the legacy Layer Properties Manager.
LAYER
Manages layers and layer properties.

PURGE
Removes unused items, such as block definitions and layers, from the drawing.

CLAYER
Sets the current layer.

**Change Layer Settings and Layer Properties**

You can change the name of a layer and any of its properties, including color and linetype, and you can reassign objects from one layer to another.

Because everything in your drawing is associated with a layer, it's likely that in the course of planning and creating a drawing, you'll need to change what you place on a layer or how you view the layers in combination. You can

- Reassign objects from one layer to another.
- Change the name of a layer.
- Change the default color, linetype, or other properties of the layer.

Reassigning an object to a different layer is useful if you create an object on the wrong layer or decide to change your layer organization. Unless the color, linetype, or other properties of the object have been set explicitly, an object that you reassign to a different layer will acquire the properties of that layer.

You can change layer properties in the Layer Properties Manager and in the Layer control on the Layers toolbar. Click the icons to change settings. Layer names and colors can be changed only in the Layer Properties Manager, not the Layer control.

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 Tools tab ➤ Drawing Utilities panel ➤ Rename.

2. In the Rename dialog box, in the Named Objects list, select Layers.

3. In Old Name, enter the old name, using wild-card characters; for example, enter stairs$*.

4. In Rename To, enter the new name using wild-card characters; for example, enter s_*. Results for this example are as follows: the layers STAIR$LEVEL-1, STAIR$LEVEL-2, STAIR$LEVEL-3 are renamed S_LEVEL-1, S_LEVEL-2, S_LEVEL-3.

5. Click 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.

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.

LAYERMANGERSTATE
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

**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.
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.
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 342 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.

<table>
<thead>
<tr>
<th>Character</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td># (pound)</td>
<td>Matches any numeric digit</td>
</tr>
<tr>
<td>@ (at)</td>
<td>Matches any alphabetic character</td>
</tr>
<tr>
<td>. (period)</td>
<td>Matches any nonalphanumeric character</td>
</tr>
<tr>
<td>* (asterisk)</td>
<td>Matches any string and can be used anywhere in the search string</td>
</tr>
<tr>
<td>? (question mark)</td>
<td>Matches any single character; for example, ?BC matches ABC, 3BC, and so on</td>
</tr>
<tr>
<td>~ (tilde)</td>
<td>Matches anything but the pattern; for example; ~<em>AB</em> matches all strings that don’t contain AB</td>
</tr>
<tr>
<td>[ ]</td>
<td>Matches any one of the characters enclosed; for example, [AB]C matches AC and BC</td>
</tr>
<tr>
<td>[~]</td>
<td>Matches any character not enclosed; for example, [~AB]C matches XC but not AC</td>
</tr>
<tr>
<td>[-]</td>
<td>Specifies a range for a single character; for example, [A-G]C matches AC, BC, and so on to GC, but not HC</td>
</tr>
<tr>
<td>` (reverse quote)</td>
<td>Reads the next character literally; for example, `~AB matches ~AB</td>
</tr>
</tbody>
</table>
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 342

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.
MAXSORT
  Sets the maximum number of symbol names or block names sorted by listing commands.
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 342 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.

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.

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.

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, linewidth, 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 Name$0$Layer 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 LT Color Index (ACI)
- True Color®, PANTONE®
- RAL™ Classic and RAL Design color books
- DIC® Color Guide
- Colors from imported color books.

**ACI Colors**

ACI colors are the standard colors used in AutoCAD LT. Each color is identified by an ACI number, an integer from 1 through 255. Standard color names are available only for colors 1 through 7. The colors are assigned as follows: 1 Red, 2 Yellow, 3 Green, 4 Cyan, 5 Blue, 6 Magenta, 7 White/Black.

**True Colors**

True colors use 24-bit color definitions to display over 16 million colors. When specifying true colors, you can use either an RGB or HSL color model. With the RGB color model, you can specify the red, green, and blue components of the color; with the HSL color model, you can specify the hue, saturation, and luminance aspects of the color.

**Color Books**

This program includes several standard Pantone color books. You can also import other color books such as the DIC color guide or RAL color sets. Importing user-defined color books can further expand your available color selections.

**NOTE** Pantone has provided new color definitions for Architectural & Interiors Cotton and Architectural & Interiors Paper color books. If you used these color books in releases prior to AutoCAD 2006, you may notice subtle changes in the colors.

You install color books on your system by using the Files tab in the Options dialog box. Once a color book is loaded, you can select colors from the color book and apply them to objects in your drawings.
All objects are created using the current color, which is displayed in the Color control on the Properties toolbar. You can also set the current color with the Color control or the Select Color dialog box.

If the current color is set to BYLAYER, objects are created with the color assigned to the current layer. If you do not want the current color to be the color assigned to the current layer, you can specify a different color.

If the current color is set to BYBLOCK, objects are created using color 7 (white or black) until the objects are grouped into a block. When the block is inserted into the drawing, it acquires the current color setting.

**To set an ACI color for all new objects**

1. Click Home tab ➤ Properties panel ➤ Object Color.
2. In the Object Color drop-down list, click a color to draw all new objects in that color, or click Select Color to display the Select Color dialog box and do one of the following:
   - On the Index Color tab, click a color or enter the color name or number in the Color box.
   - On the Index Color tab, click BYLAYER to draw new objects in the color assigned to the current layer.
   - On the Index Color tab, click BYBLOCK to draw new objects in the current color until they are grouped into a block. When the block is inserted into the drawing, the objects in the block acquire the current color setting.
3. Click OK.
   The Color control displays the current color.

**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.
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.

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 current color setting on the Properties toolbar from BYLAYER to a specific color.

See also:

■ Override Layer Properties in Viewports on page 325

**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.

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.

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 LT 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.

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You can define a custom linetype that will orient the imbedded text to keep it readable automatically.

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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, polylnes, splines, and helixes, which
is useful for linetypes with included text, or wide polylnes 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 list. 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, which is displayed in the Linetype control on the Properties toolbar.

You can also set the current linetype with the Linetype control.
If the current linetype is set to BYLAYER, objects are created with the linetype assigned to the current layer.
If the current linetype is set to BYBLOCK, objects are created using the CONTINUOUS linetype until the objects are grouped into a block. When the block is inserted into the drawing, those objects acquire the current linetype setting.
If you do not want the current linetype to be the linetype assigned to the current layer, you can specify a different linetype explicitly.
The program does not display the linetype of certain objects: text, points, viewports, hatches, and blocks.

To set the linetype for all new objects

1 Click Home tab ➤ Properties panel ➤ Linetype.

2 In the Linetype drop-down list, click Other. Then, in the Linetype Manager dialog box, click Load.
   You can hold down Ctrl to select several linetypes or SHIFT to select a range of linetypes.

3 In the Linetype Manager dialog box, do one of the following:
   ■ Select a linetype and select Current to draw all new objects with that linetype.
   ■ Select BYLAYER to draw new objects in the linetype assigned to the current layer.
   ■ Select BYBLOCK to draw new objects in the current linetype until they are grouped into a block. When the block is inserted into a drawing, the objects in the block acquire the current linetype setting.
Click Home tab ➤ Layers panel ➤ Layer Properties.

In the Layer Properties Manager, select the linetype name you want to change.

In the Select Linetype dialog box, select the linetype you want. Click OK.

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 current linetype setting on the Properties toolbar from BYLAYER to a specific linetype.

See also:
- Override Layer Properties in Viewports on page 325

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.
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

LINETYPE
Loads, sets, and modifies linetypes.

CELTSCALE
Sets the current object linetype scaling factor.

LTSCALE

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 367.

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.

Display Linetypes on Short Segments and Polylines | 369
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 LWT button on the status bar is selected.

TrueType fonts, raster images, points, and solid fills (2D solids) cannot display lineweight. Wide polylines show lineweights only when displayed outside of the plan view. You can export drawings to other applications or cut objects to the Clipboard and retain lineweight information.

In model space, lineweights are displayed in pixels and do not change when zoomed in or out. Thus, you should not use lineweights to represent the exact width of an object in model space. For example, if you want to draw an object with a real-world width of 0.5 inches, do not use a lineweight; instead, use a polyline with a width of 0.5 inches to represent the object.

You can also plot objects in your drawing with custom lineweight values. Use the Plot Style Table Editor to adjust the fixed lineweight values to plot at a new value.

### Lineweight Scale in Drawings

Objects with a lineweight are plotted with the exact width of the assigned lineweight value. The standard settings for these values include BYLAYER, BYBLOCK, and Default. They are displayed in either inches or millimeters, with millimeters being the default. All layers are initially set to 0.25 mm, controlled by the LWDEFAULT system variable.

A lineweight value of 0.025 mm or less is displayed as one pixel in model space and is plotted at the thinnest lineweight available on the specified plotting device. Lineweight values that you enter at the Command prompt are rounded to the nearest predefined value.

You set the lineweight units and the default value in the Lineweight Settings dialog box. You can access the Lineweight Settings dialog box by using the LWEIGHT command, by right-clicking the LWT button on the status bar and choosing Settings, or by choosing Lineweight Settings on the User Preferences tab in the Options dialog box.

**See also:**

- Draw Polylines on page 475

### 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.
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 LWT on the status bar. This setting does not affect the plotting of lineweights.

Display Lineweights in Model Space
Lineweight display in model space does not change with the zoom factor. For example, a lineweight value that is represented by a width of four pixels is always displayed using four pixels regardless of how far you zoom in. If you want the lineweights on objects to appear thicker or thinner on the Model tab, use LWEIGHT to set their display scale. Changing the display scale does not affect the lineweight plotting value.

In model space, weighted lines that are joined form a beveled joint with no end caps. You can use plot styles to apply different joins and endcap styles to objects with lineweights.

NOTE Different styles of endcaps and joins of objects with lineweight are displayed only in a full plot preview.
Display Lineweights in Layouts

In layouts and plot preview, lineweights are displayed in real-world units, and lineweight display changes with the zoom factor. You can control lineweight plotting and scaling in your drawing in the Plot dialog box, Plot Settings tab.

To display or hide lineweights

Use one of the following methods:

- Click LWT on the status bar.
- Select or clear Display Lineweight in the Lineweight Settings dialog box.
- Set the LWDISPLAY system variable to 0 or 1.

Quick Reference

LAYER
Manages layers and layer properties.

LWEIGHT
Sets the current lineweight, lineweight display options, and lineweight units.

PEDIT
Edits polylines.

PLINE
Creates a 2D polyline.

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, which is displayed in the Lineweight control on the Properties toolbar. You can also set the current lineweight with the Lineweight control.

If the current lineweight is set to BYLAYER, objects are created with the lineweight assigned to the current layer.

If the current lineweight is set to BYBLOCK, objects are created using the default lineweight setting until the objects are grouped into a block. When the block is inserted into the drawing, it acquires the current lineweight setting.

If you do not want the current lineweight to be the lineweight assigned to the current layer, you can specify a different lineweight explicitly.

Objects in drawings created in an earlier release of AutoCAD LT 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.
PLINE
Creates a 2D polyline.
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 current lineweight setting on the Properties toolbar from BYLAYER to a specific lineweight.

See also:
- Override Layer Properties in Viewports on page 325

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 \texttt{w} (Width) and enter a new width for all segments.
4. Press Enter to end the command.

Quick Reference

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<td>LWDEFAULT</td>
<td>Sets the value for the default lineweight.</td>
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<tr>
<td>LWDISPLAY</td>
<td>Controls whether the lineweights of objects are displayed.</td>
</tr>
</tbody>
</table>
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.

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.
Turn Off Lineweights

Any lineweight width that is represented by more than one pixel may slow down performance. If you want to improve display performance, turn lineweights off. You can turn lineweights on and off by choosing the LWT button on the status bar or by using the Lineweight Settings dialog box. Lineweights are always plotted at their real-world value whether their display is turned on or off.

Update the Display

New objects automatically use the current settings for displays of solid fill and text. Except for lineweights, to update the display of existing objects using these settings, you must use REGEN.

See also:
- Use Layers to Manage Complexity on page 313
- Display Lineweights on page 373
- Use TrueType Fonts on page 905

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.

![Image showing transparency settings]

**IMPORTANT** For performance reasons, plotting transparency is disabled by default. To plot transparent objects, check the Plot Transparency option in either the Plot dialog box or the 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 Transparency 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.

CETRANSPERACY
Sets the transparency level for new objects.

TRANSPERCENCYDISPLAY
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.

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
Writes objects or a block to a new 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. A light bulb icon on the status bar indicates when one or more objects are isolated or hidden. To redisplay objects, use UNISOLATEOBJECTS. 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.

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

You can use a variety of precision drawing tools to help you produce accurate drawings quickly and without performing tedious calculations.

Use Coordinates and Coordinate Systems (UCS)

For precise coordinate input, you can use several coordinate system entry methods. You can also use a movable coordinate system, the user coordinate system (UCS), for convenient coordinate entry and to establish workplanes.

Overview of Coordinate Entry

When a command prompts you for a point, you can use the pointing device to specify a point, or you can enter a coordinate value at the Command prompt. When dynamic input is on, you can enter coordinate values in tooltips near the cursor. You can enter two-dimensional coordinates as either Cartesian ($X, Y$) or polar coordinates.

Cartesian and Polar Coordinates

A Cartesian coordinate system has three axes, $X$, $Y$, and $Z$. When you enter coordinate values, you indicate a point’s distance (in units) and its direction (+ or -) along the $X$, $Y$, and $Z$ axes relative to the coordinate system origin (0,0,0).

In 2D, you specify points on the $XY$ plane, also called the workplane. The workplane is similar to a flat sheet of grid paper. The $X$ value of a Cartesian coordinate specifies horizontal distance, and the $Y$ value specifies vertical distance. The origin point (0,0) indicates where the two axes intersect.

Polar coordinates use a distance and an angle to locate a point. With both Cartesian and polar coordinates, you can enter absolute coordinates based on the origin (0,0), or relative coordinates based on the last point specified.
Another method of entering a relative coordinate is by moving the cursor to specify a direction and then entering a distance directly. This method is called direct distance entry.

You can enter coordinates in scientific, decimal, engineering, architectural, or fractional notation. You can enter angles in grads, radians, surveyor’s units, or degrees, minutes, and seconds. The UNITS command controls unit format.

**Display Coordinates on the Status Bar**

The current cursor location is displayed as a coordinate value on the status bar.

There are three types of coordinate display: static, dynamic, and distance and angle.

- **Static display.** Updates only when you specify a point.
- **Dynamic display.** Updates as you move the cursor.
- **Distance and angle display.** Updates the relative distance \(\text{distance-angle}\) 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 390
- [Enter Polar Coordinates](#) on page 392
- [Enter 3D Coordinates](#) on page 394
- [Use Dynamic Input](#) on page 406

**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.
   If the BLIPMODE system variable is on, a blip (a small cross) is displayed at the point location.

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

*BLIPMODE*
- Controls the display of marker blips.

*ID*
- Displays the UCS coordinate values of a specified location.

*LIST*
- Displays property data for selected objects.

*BLIPMODE*
- Controls the display of marker blips.

*COORDS*
- Controls the format and update frequency of coordinates on the status line.

*LASTPOINT*
- Stores the last point specified, expressed as UCS coordinates for the current space.
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 406.

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:

![Diagram of a line drawn with Cartesian coordinates]

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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

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 406.

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:
  
  \[
  \text{@distance<angle}
  \]

Quick Reference

UNITS

Controls coordinate and angle display formats and precision.

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
GRID
  Displays a grid pattern in the current viewport.
SNAP
  Restricts cursor movement to specified intervals.
TABLET
  Calibrates, configures, and turns on and off an attached digitizing tablet.
UCS
  Manages user coordinate systems.
UCSICON
  Controls the visibility and placement of the UCS icon.
UNITS
  Controls coordinate and angle display formats and precision.
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:

\[ \mathbf{X} < \text{angle from X axis}, \mathbf{Z} \]

**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.

![Cylindrical Coordinates Diagram](image)

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 current point.
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:
  
  @<angle from the $X$ axis>,
  
  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**

**UNITS**

Controls coordinate and angle display formats and precision.

**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:
  \[@x<\text{angle from the } x \text{ axis}<\text{angle from the } xy \text{ plane}\]

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

UNITs

Controls coordinate and angle display formats and precision.

COORDS

Controls the format and update frequency of coordinates on the status line.

Understand the User Coordinate System (UCS)

You can relocate and rotate the user coordinate system for convenient coordinate entry, grid display, grid snap, Ortho mode, and other drawing tools.
Understand the World and User Coordinate Systems

There are two coordinate systems: a fixed system called the world coordinate system (WCS) and a movable system called the user coordinate system (UCS). By default, these two systems are coincident in a new drawing.

Normally in 2D views, the WCS $X$ axis is horizontal and the $Y$ axis is vertical. The WCS origin is where the $X$ and $Y$ axes intersect (0,0). All objects in a drawing file are defined by their WCS coordinates. However, it is usually more convenient to create and edit objects based on the movable UCS.

Work with the User Coordinate System

Virtually all coordinate entry as well as many other tools and operations reference the current UCS. 2D tools and operations that depend on the location and orientation of the UCS include the following:

- Absolute and relative coordinate entry
- Absolute reference angles
- Definition of horizontal and vertical for Ortho mode, polar tracking, object snap tracking, grid display, and grid snap
- Orientation of horizontal and vertical dimensions
- Orientation of text objects
- View rotation using the PLAN command

Moving or rotating the UCS can make it easier to work on particular areas of a drawing.

You can relocate the user coordinate system with methods such as the following:

- Move the UCS by defining a new origin point.
- Align the UCS with an existing object.
- Rotate the UCS by specifying a new origin point and a point on the new $X$ axis.
- Rotate the current UCS a specified angle around the $Z$ axis.
- Revert to the previous UCS.
- Restore the UCS to be coincident with the WCS.
Each of these methods have a corresponding option in the UCS command. Once you have defined a UCS, you can name it and then restore it when you need to use it again.

**To define a new UCS origin in 2D**

1. Click View tab ➤ Coordinates panel ➤ Origin.
2. Specify a point for the new origin. The UCS origin (0,0) is redefined at the point you specify.

**To change the rotation angle of the UCS**

1. Click View tab ➤ Coordinates panel ➤ Z.
2. Specify a rotation angle.

**To restore the UCS to be coincident with the WCS**

1. Click View tab ➤ Coordinates panel ➤ Named UCS.
2. In the UCS dialog box, Named UCSs tab, select World.
3. Click Set Current.
4. Click OK.

**To restore the previous UCS**

1. Click View tab ➤ Coordinates panel ➤ Named UCS.
2. In the UCS dialog box, Named UCSs tab, select Previous.
3. Click Set Current.
4. Click OK.

**To save a UCS**

1. Click View tab ➤ Coordinates panel ➤ Named UCS. The new UCS is displayed in the UCS list as UNNAMED.
2 In the UCS dialog box, Named UCSs tab, select UNNAMED and enter a new name. (You can also select UNNAMED, and right-click. Click Rename.)

3 Click OK.

You can use up to 255 characters, including letters, digits, and the special characters dollar sign ($), hyphen (-), and underscore (_). All UCS names are converted to uppercase.

To restore a named UCS

1 Click View tab ➤ Coordinates panel ➤ Named UCS.

2 In the UCS dialog box, Named UCSs tab, you can view the origin and axis direction of a listed UCS. Select the UCS name. Click Details. When you have finished viewing the list, click OK to return to the UCS dialog box.

3 Select the coordinate system you want to restore. Click Set Current.

4 Click OK.

To rename a UCS

1 Click View tab ➤ Coordinates panel ➤ Named UCS.

2 In the UCS dialog box, Named UCSs tab, select the coordinate system you want to rename. (You can also select UNNAMED, and right-click. Click Rename.)

3 Enter a new name.

4 Click OK.

To delete a named UCS

1 Click View tab ➤ Coordinates panel ➤ Named UCS.

2 In the UCS dialog box, Named UCSs tab, select the UCS you want to delete.

3 Press Delete.

You cannot delete the current UCS or a UCS with the default name UNNAMED.
Quick Reference

UCS
Manages user coordinate systems.

UCSICON
Controls the visibility and placement of the UCS icon.

UCSMAN
Manages defined user coordinate systems.

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 Display of the User Coordinate System Icon

To help visualize the current orientation of the user coordinate system, you can display the user coordinate system icon. Several versions of this icon are available, and you can change its size, location, and color.

To indicate the location and orientation of the UCS, the UCS icon is displayed either at the UCS origin point or in the lower-left corner of the current viewport.

You can choose 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.

Use the UCSICON command to choose between displaying the 2D or the 3D UCS icon. The shaded UCS icon is displayed when you open a drawing with a shaded 3D view that was created in AutoCAD. To indicate the origin and orientation of the UCS, you can display the UCS icon at the UCS origin point using the UCSICON command.

If you have multiple viewports, each viewport displays its own UCS icon.

The UCS icon is displayed in various ways to help you visualize the orientation of the workplane. The following figure shows some of the possible icon displays.
You can use the UCSICON command to switch between the 2D UCS icon and the 3D UCS icon. You can also use the command to change the size, color, and icon line width of the 3D UCS icon.

The UCS broken pencil icon replaces the 2D UCS icon when the viewing direction is in a plane parallel to the UCS $XY$ plane. The broken pencil icon indicates that the edge of the $XY$ plane is almost perpendicular to your viewing direction. This icon warns you not to use your pointing device to specify coordinates.

When you use the pointing device to locate a point, it's normally placed on the $XY$ plane. If the UCS is rotated so that the $Z$ axis lies in a plane parallel to the viewing plane—that is, if the $XY$ plane is edge-on to the viewer—it may be difficult to visualize where the point will be located. In this case, the point will be located on a plane parallel to your viewing plane that also contains the UCS origin point. For example, if the viewing direction is along the $X$ axis, coordinates specified with a pointing device will be located on the $YZ$ plane, which contains the UCS origin point.

Use the 3D UCS icon to help you visualize which plane these coordinates will be projected on; the 3D UCS icon does not use a broken pencil icon.

**To turn the display of the UCS icon on and off**

- Click View tab ➤ 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

UCSICON

Controls the visibility and placement of the UCS icon.

UCSICON

Displays the UCS icon for the current viewport or layout.

Use Dynamic Input

Dynamic Input provides a command interface near the cursor to help you keep your focus in the drafting area.

When dynamic input is on, tooltips display information near the cursor that is dynamically updated as the cursor moves. When a command is active, the tooltips provide a place for user entry.

After you type a value in an input field and press Tab, the field then displays a lock icon, and the cursor is constrained by the value that you entered. You can then enter a value for the second input field. Alternately, if you type a value and press Enter, the second input field is ignored and the value is interpreted as direct distance entry.

The actions required to complete a command or to use grips are similar to those for the Command prompt. The difference is that your attention can stay near the cursor.

Dynamic input is not designed to replace the command window. You can hide the command window to add screen area for drawing, but you will need to display it for some operations. Press F2 to hide and display Command prompts and error messages as needed. Alternately, you can undock the command window and use Auto-hide to roll open or roll up the window.

Turn On or Turn Off Dynamic Input

Click the dynamic input button \[\text{on the status bar}\] to turn dynamic input on and off. You can turn it off temporarily by holding down the F12 key. Dynamic input has three components: pointer input, dimensional input, and dynamic prompts. Right-click \[\text{and click Settings}\] to control what is displayed by each component when dynamic input is on.
**Pointer Input**

When pointer input is on and a command is active, the location of the crosshairs is displayed as coordinates in a tooltip near the cursor. You can enter coordinate values in the tooltip instead of on the command line.

The default for second and subsequent points is relative polar coordinates (relative Cartesian for RECTANG). There is no need to type the at sign (@). If you want to use absolute coordinates, use the pound sign (#) prefix. For example, to move an object to the origin, for the second point prompt, enter #0,0.

Use the pointer input settings to change the default format for coordinates and to control when pointer input tooltips are displayed.

**Dimensional Input**

When dimensional input is on, the tooltips display distance and angle values when a Command prompts for a second point. The values in the dimensional tooltips change as you move the cursor. Press Tab to move to the value you want to change. Dimensional input is available for ARC, CIRCLE, ELLIPSE, LINE, and PLINE.

When you use grips to edit an object, the dimensional input tooltips can display the following information:

- Original length
- A length that updates as you move the grip
- The change in the length
- Angle
- The change in the angle as you move the grip
- The radius of an arc

Use the dimensional input settings to display only the information you want to see.

When you use grips to stretch objects or when you create new objects, dimensional input displays only acute angles, that is, all angles are displayed as 180 degrees or less. Thus, an angle of 270 degrees is displayed as 90 degrees regardless of the ANGDIR system variable setting (set in the Drawing Units dialog box). Angles specified when creating new objects rely on the cursor location to determine the positive angle direction.

**Dynamic Prompts**

When dynamic prompts are on, prompts are displayed in a tooltip near the cursor. You can enter a response in the tooltip instead of on the command line. Press the DOWN ARROW key to view and select options. Press the UP ARROW key to display recent input.

**NOTE** To use PASTECLIP in a dynamic prompt tooltip, type a letter and then backspace to delete it before you paste the entry. Otherwise, the entry is pasted into the drawing as text.

**To enter coordinate values in dynamic input tooltips**

1. On the status bar, verify that the dynamic input button is on.
2 Use one of the following methods to enter coordinate values or select options:

- To enter polar coordinates, enter the distance from the first point and press Tab, and then enter an angle value and press Enter.

- To enter Cartesian coordinates, enter an X coordinate value and a comma (,), and then enter a Y coordinate value and press Enter.

- If a down-arrow icon follows the prompt, press the DOWN ARROW key until a dot is displayed next to the option. Press Enter.

- Press the UP ARROW key to access recent coordinates, or right-click and click Recent Input to access the coordinates from a shortcut menu.

**NOTE** For dimensional input, after you type a value in an input field and press Tab, the field then displays a lock icon, and the cursor is constrained by the value that you entered.

**To correct typing errors in dynamic input tooltips**

- When a dynamic input tooltip displays the red error outline, the current entry is selected. Type over the selected text to replace it. You can also use the RIGHT ARROW, LEFT ARROW, Backspace, and Delete keys to correct your entry. After you make the correction, press Tab, comma (,), or a left angle bracket (<) to remove the red outline and complete the coordinates.

- If you type the @ or # or * prefixes in a pointer input tooltip and then want to change it, you can just type the character you want. There is no need to backspace.

**To specify relative or absolute coordinates in pointer input tooltips**

- To enter absolute coordinates when relative coordinates are displayed in the tooltip, enter # to temporarily override the DYNPICOORDS system variable.

- To enter relative coordinates when absolute coordinates are displayed, enter @ to temporarily override the DYNPICOORDS system variable.

- To enter absolute world coordinate system (WCS) coordinates, enter *.

**NOTE** During pointer input, you can use the shortcut menu to access the # and * prefixes.
To 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**

DSETTINGS

Sets grid and snap, polar and object snap tracking, object snap modes, Dynamic Input, and Quick Properties.

Use Dynamic Input | 411
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
- Click an object snap button on the Object Snap toolbar
- Enter the name of an object snap at the Command prompt
- On the status bar, right-click the object snap button

When you specify an object snap at a prompt for a point, the object snap stays in effect only for the next point that you specify.

NOTE Object snaps work only when you are prompted for a point. If you try to use an object snap at the Command prompt, an error message is displayed.

Use Running Object Snaps

If you need to use one or more object snaps repeatedly, you can turn on running object snaps. For example, you might set Center as a running object snap if you need to connect the centers of a series of circles with a line.

You can specify one or more running object snaps on the Object Snaps tab in the Drafting Settings dialog box, which is accessible from the Tools menu. If several running object snaps are on, more than one object snap may be eligible
at a given location. Press Tab to cycle through the possibilities before you specify the point.

Click the OSNAP button on the status bar or press F3 to turn running object snaps on and off.

NOTE If you want object snaps to ignore hatch objects, set the OSOPTIONS system variable to 1.

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 button, 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

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

OPTIONS
Customizes the program settings.

OSNAP
Sets running object snap modes.

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

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

OSOPTIONS
Automatically suppresses object snaps on hatch objects.

MTP (Command Modifier)
Locates the midpoint between two points.

The Object Snap Menu

Specify an object snap quickly and conveniently from a shortcut menu.

The object snap menu is displayed at your cursor location when you hold down Shift and click the right mouse button or the equivalent button on another pointing device.

The default object snap menu lists object snaps and tracking options. If you want to change the options, you can modify a customization file. The main customization file that’s shipped with the product is acadlt.cuix.
See also:
- “Pull-down and Shortcut Menus” in the Customization Guide
- Use Object Snaps on page 413

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

**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.
   - **AutoSnap Marker Color.** Changes the color of the marker.
   - **AutoSnap Marker Size.** Adjusts the size of the marker.
3. Click OK.

**Quick Reference**

**APERTURE**

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

**OPTIONS**

Customizes the program settings.

**OSNAP**

Sets running object snap modes.

**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) |
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

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.
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 dots when SHADEMODE is set to 2D Wireframe. The grid displays lines when SHADEMODE is set to Hidden.

By default, the X and Y axes of the UCS display in a different color than the grid lines. You can control the color in the Drawing Window Colors dialog box. This dialog box is accessible from the Drafting tab in the Options dialog box.

The LIMITS command controls the drawing area covered by the grid. As an option, you can override the limits to make the grid cover the entire XY plane of the user coordinate system (UCS). You can access this option in the Drafting Settings dialog box or use the GRIDDISPLAY system variable.

**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
To turn off the display of major grid lines, set the frequency of major grid lines to 1.

NOTE If the grid is displayed as lines, the grid limits are displayed also as darker lines. Do not confuse these boundaries with major grid lines.

NOTE When the grid is displayed as lines and SNAPANG is set to a value other than 0, the grid will not display. SNAPANG does not affect the display of the dotted grid.

**Change the Grid Dynamically During Zooming**

If you zoom in or out of your drawing, the grid spacing is adjusted automatically to be more appropriate for the new magnification. This is called *adaptive grid display*.

For example, if you zoom way out, the density of displayed grid lines reduces automatically. Conversely, if you zoom way in, additional grid lines display in the same proportion as the major grid lines.
Change Grid and Snap Spacing

As you work, you can turn Grid and Snap mode on and off, and you can change the grid and snap spacing. You can turn Snap mode on and off temporarily by using an override key.

Snap spacing does not have to match grid spacing. For example, you might set a wide grid spacing to be used as a reference but maintain a closer snap spacing for accuracy in specifying points.

Change the Grid and Snap Angle and Base

If you need to draw along a specific alignment or angle, you can change the grid and snap angle by rotating the user coordinate system (UCS). This rotation realigns the crosshairs on the screen to match the new angle. In the following example, the UCS is rotated 30 degrees to match the angle of the anchor bracket.

The grid and snap points are always aligned with the UCS origin. If you need to shift the grid and grid snap origin, move the UCS.

See also:

■ "Set Isometric Grid and Snap"

■ Override Object Snap Settings on page 418

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 SHADEMODE.
2 Do one of the following:
   ■ To display the grid as dots, specify the 2D Wireframe option.
   ■ To display the grid as lines, specify the Hidden option.

To change the frequency of major grid lines
1 If necessary, at the Command prompt, enter SHADEMODE and specify the Hidden visual style.
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

DSETTINGS
Sets grid and snap, polar and object snap tracking, object snap modes, Dynamic Input, and Quick Properties.

GRID
Displays a grid pattern in the current viewport.

LIMITS
Sets and controls the limits of the grid display in the current Model or layout tab.

SHADEMODE
Controls the display of solid object shading in the current viewport.

SNAP
Restricts cursor movement to specified intervals.
GRIDDISPLAY
  Controls the 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 428.

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 418

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

ORTHO
Constrains cursor movement to the horizontal or vertical direction.

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.

Use PolarSnap™ to snap to specified distances along the alignment path. For example, in the following illustration you draw a two-unit line from point 1 to point 2, and then draw a two-unit line to point 3 at a 45-degree angle to the line. If you turn on the 45-degree polar angle increment, an alignment path and tooltip are displayed when your cursor crosses the 0 or 45-degree angle. The alignment path and tooltip disappear when you move the cursor away from the angle.
As you move your cursor, alignment paths and tooltips are displayed when you move the cursor near polar angles. The default angle measurement is 90 degrees. Use the alignment path and tooltip to draw your object. You can use polar tracking with Intersection and Apparent Intersection object snaps to find where a polar alignment path intersects another object.

**NOTE** Ortho mode and polar tracking cannot be on at the same time. Turning on polar tracking turns off Ortho mode. Similarly, PolarSnap and grid snap cannot be on at the same time. Turning on PolarSnap turns off grid snap.

### Specify Polar Angles (Polar Tracking)

You can use polar tracking to track along polar angle increments of 90, 60, 45, 30, 22.5, 18, 15, 10, and 5 degrees, or you can specify other angles. The following illustration shows the alignment paths displayed as you move your cursor 90 degrees with the polar angle increment set to 30 degrees.

The orientation of 0 depends on the angle you set in the Drawing Units dialog box (UNITS). The direction of snap (clockwise or counterclockwise) depends on the units direction you specify when setting units of measurement.

You can turn polar tracking on and off temporarily by using an override key. The direct distance entry method is not available while you are using the temporary override key for polar tracking.

### Specify Polar Distances (PolarSnap)

PolarSnap restricts cursor movement to increments of a polar distance you specify. For example, if you specify a length of 4 units, the cursor snaps from
the first point specified to lengths of 0, 4, 8, 12, 16, and so on. As you move your cursor, a tooltip indicates the nearest PolarSnap increment. To restrict point entry to polar distances, both polar tracking and Snap mode (set to PolarSnap) must be on. You can turn off all snapping and tracking temporarily by using an override key.

See also:

■ Override Object Snap Settings on page 418

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

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.

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.
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.

Quick Reference

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 418.

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).

![Object Snap Tracking Illustration](image)

**Change Object Snap Tracking Settings**

By default, object snap tracking is set to orthogonal. Alignment paths are displayed at 0, 90, 180, and 270 degrees from acquired object points. However, you can use polar tracking angles instead.

For object snap tracking, object points are automatically acquired. However, you can choose to acquire points only when you press Shift.

**Change Alignment Path Display**

You can change how AutoTrack displays alignment paths, and you can change how object points are acquired for object snap tracking. By default, alignment paths stretch to the end of the drawing window. You can change their display to abbreviated lengths, or no length.
Tips for Using Object Snap Tracking

As you use AutoTrack (polar tracking and object snap tracking), you will discover techniques that make specific design tasks easier. Here are a few you might try.

- Use Perpendicular, End, and Mid object snaps with object snap tracking to draw to points that are perpendicular to the end and midpoints of objects.

- Use the Tangent and End object snaps with object snap tracking to draw to points that are tangent to the endpoints of arcs.

- Use object snap tracking with temporary tracking points. At a point prompt, enter tt, then specify a temporary tracking point. A small + appears at the point. As you move your cursor, AutoTrack alignment paths are displayed relative to the temporary point. To remove the point, move the cursor back over the +.

- After you acquire an object snap point, use direct distance to specify points at precise distances along alignment paths from the acquired object snap point. To specify a point prompt, select an object snap, move the cursor to display an alignment path, then enter a distance at the Command prompt.

**NOTE** The direct distance entry method is not available while you are using the temporary override key for object snap tracking.

- Use the Automatic and Shift to Acquire options set on the Drafting tab of the Options dialog box to manage point acquisition. Point acquisition is set to Automatic by default. When working in close quarters, press Shift to temporarily avoid acquiring a point.

To turn on and turn off object snap tracking

- Press F11, or click on the status bar.
  To turn object snap tracking on and off temporarily, hold down the F11 key while you work.

To change AutoTrack settings

1. Click Tools menu ➤ Options.
2 In the Options dialog box, Drafting tab, under AutoTrack Settings, select or clear the following alignment path display options:

- **Display Polar Tracking Vector.** Controls alignment path display for object snap tracking. When cleared, no polar tracking path is displayed.

- **Display Full Screen Tracking Vector.** Controls alignment path display for object snap tracking. When cleared, an alignment path is displayed only from the object snap point to the cursor.

- **Display AutoTrack Tooltip.** Controls the display of AutoTrack tooltips. Tooltips tell you the type of object snap (for object snap tracking), alignment angle, and distance from the previous point.

3 Under Alignment Point Acquisition, select a method for acquiring object points for object snap tracking:

- **Automatic.** Acquires object points automatically. If you select this option, you can press Shift to not acquire an object point

- **Shift to Acquire.** Acquires object points only when you press Shift while the cursor is over an object snap point.

**Quick Reference**

**DSETTINGS**
Sets grid and snap, polar and object snap tracking, object snap modes, Dynamic Input, and Quick Properties.

**OPTIONS**
Customizes the program settings.

**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

TRACKING (Command Modifier)

Locates a point from a series of temporary points.

Specify Distances

When specifying a point, you can enter distances, offsets, and measured intervals.

Enter Direct Distances

You can specify a point by moving the cursor to indicate a direction and then entering the distance.

To specify a line length quickly, without entering coordinate values, you can specify a point by moving the cursor to indicate a direction and then entering the distance from the first point. You can enter calculated distances from the QuickCalc calculator. For more information, see Use the QuickCalc Calculator on page 454.

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 428
- Lock an Angle for One Point (Angle) on page 432
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

LINE
Creates straight line segments.

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

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 655
Quick Reference

BLOCK
Creatcs a block definition from selected objects.

DDPTYPE
Specifies the display style and size of point objects.

DIVIDE
Creates evenly spaced point objects or blocks along the length or perimeter of an object.

MEASURE
Creates point objects or blocks at measured intervals along the length or perimeter of an object.

WBLOCK
Writes objects or a block to a new drawing file.

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 using several methods. To change the point style in a dialog
box, you can use DDPTYPE. Alternately, click Format menu ➤ Point Style. The PDMODE system variable also controls the appearance of point markers. For example, you can change the value to make points appear as crosses. PDSIZE controls the size of point objects.

To insert points at measured intervals on an object

1. Click Home tab ➤ Draw panel ➤ Point drop-down ➤ Measure.
2. Select a line, arc, spline, circle, ellipse, or polyline.
3. Enter an interval length, or specify points to indicate a length. Points are placed on the object at the specified intervals.

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

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.

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 using several methods. To change the point style in a dialog box, you can use DDPTYPE. Alternately, click Format menu ➤ Point Style. The PDMODE system variable also controls the appearance of point markers. For example, you can change the value to make points appear as crosses. PDSIZE controls the size of point objects.
To insert points to mark equal segments

1. Click Home tab ➤ Draw panel ➤ Point drop-down ➤ Divide.
2. Select a line, circle, ellipse, polyline, arc, or spline.
3. Enter the number of segments you want.
   A point is placed between each segment.

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

BLOCK
Creates a block definition from selected objects.

DDPTYPE
Specifies the display style and size of point objects.

DIVIDE
Creates evenly spaced point objects or blocks along the length or perimeter of an object.

WBLOCK
Writes objects or a block to a new drawing file.
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 on page 387
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
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.
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 QuickCalc calculator to convert from one system of area units to another. For more information, see Use the QuickCalc Calculator on page 454.
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 512
- Overview of Object Properties on page 305

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

AREA
LIST
Displays property data for selected objects.

MASSPROP
Calculates the mass properties of regions or AutoCAD 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.

AREA
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 Command prompt calculator.

Use the QuickCalc Calculator
With the QuickCalc calculator, an interface that looks and functions like a hand-held calculator, you can perform mathematical, scientific, and geometric calculations, convert units of measurement, manipulate the properties of objects, and evaluate expressions.

Overview of QuickCalc
The QuickCalc calculator includes basic features similar to most standard mathematical calculators.

In addition, QuickCalc has features specific to AutoCAD LT 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 107.

Quick Reference
QUICKCALC
Opens the QuickCalc calculator.
QCLOSE
Closes the QuickCalc calculator.

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

QUICKCALC
Opens the QuickCalc calculator.
QCCLOSE
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

QUICKCALC
Opens the QuickCalc calculator.

QCCLOSE
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.

Use the QuickCalc Calculator | 463
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

QUICKCALC
Opens the QuickCalc calculator.
QCCLOSE
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.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Shortcut For</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>dee</td>
<td>dist(end,end)</td>
<td>Distance between two endpoints</td>
</tr>
<tr>
<td>ille</td>
<td>ill(end,end,end)</td>
<td>Intersection of two lines defined by four endpoints</td>
</tr>
<tr>
<td>mee</td>
<td>(end+end)/2</td>
<td>Midpoint between two endpoints</td>
</tr>
<tr>
<td>nee</td>
<td>nor(end,end)</td>
<td>Unit vector in the XY plane and normal to two endpoints</td>
</tr>
<tr>
<td>rad</td>
<td>rad</td>
<td>Radius of a selected circle, arc, or polyline arc</td>
</tr>
<tr>
<td>vee</td>
<td>vee(end,end)</td>
<td>Vector from two endpoints</td>
</tr>
<tr>
<td>vee1</td>
<td>vec1(end,end)</td>
<td>Unit vector from two endpoints</td>
</tr>
</tbody>
</table>

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.

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.

Quick Reference

QCCLOSE
Closes the QuickCalc calculator.

QUICKCALC
Opens the QuickCalc calculator.

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 LT 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**

<table>
<thead>
<tr>
<th>Operator</th>
<th>Operation</th>
</tr>
</thead>
<tbody>
<tr>
<td>(</td>
<td>Groups expressions</td>
</tr>
<tr>
<td>^</td>
<td>Indicates numeric exponent</td>
</tr>
<tr>
<td>*, /</td>
<td>Multiplies and divides numbers</td>
</tr>
<tr>
<td>+, -</td>
<td>Adds and subtracts numbers</td>
</tr>
</tbody>
</table>

**Calculating Points**

You can use CAL whenever you need to calculate a point or a number within a command.

For example, you enter \((\text{mid}+\text{cen})/2\) 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 prompt 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

CAL
Evaluator mathematical and geometric expressions.

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

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 305.

You specify the locations that define the endpoints of each line with precision. You can

- Enter the coordinate values for an endpoint, using either absolute or relative coordinates
- Specify an object snap relative to an existing object. For example, you can specify the center of a circle as one endpoint of the line
- Turn grid snap on and snap to a location

There are other methods for creating precise lines. A highly efficient technique is to offset a line from an existing line, and then trim or extend it to the desired length.
Use polyline objects instead of line objects if you want the segments to be connected as a single object.

See also:

- Use Coordinates and Coordinate Systems (UCS) on page 387
- Use Object Snaps on page 413
- Adjust Grid and Grid Snap on page 420
- Draw Polylines on page 475
- Offset an Object on page 582
- Break and Join Objects on page 606

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 circuit board layouts
- Process and piping diagrams

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.

![Varying and uniform width](image)

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.

![Internal point and boundary](image)

See also:

- [Draw Rectangles and Polygons](#) on page 480
- [Modify Complex Objects](#) on page 608
- [Break and Join Objects](#) on page 606
- [Control Lineweights](#) on page 370
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.

PLINE
  Creates a 2D polyline.

POLYGON
  Creates an equilateral closed polyline.

RECTANG
  Creates a rectangular polyline.

System Variables

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

HPBOUND
  Controls the object type created by 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 475

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.
Specify the first corner of the rectangle.
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 Double Lines

Each double-line segment and its endcap, a line that connects the double-line endpoints, is actually a separate line object that can be individually edited.

Double lines provide a good way to draw walls in floor plans. Each double-line segment and its endcap, a line that connects the double-line endpoints, is actually a separate line object that can be individually edited.

You can draw double lines as straight segments or as arcs. The width determines the amount of distance between the lines. You can specify the width before you draw the double line or after you specify the first point.

Control Placement of Double Lines

The dragline is a rubber-band line that determines the placement of a double line. You can control whether the double line is centered on or offset to the right or left of the dragline.

A negative offset value positions the offset side of the double line to the left of the specified points; a positive value positions the offset side to the right. Determine the left and right by imagining that you are standing at the start point of the line and looking at the endpoint.

Set the Appearance of Endpoints

The ends of double lines can be open, closed, or different on each end. When endcaps are turned on, the program connects, or caps, the specified ends. The Auto option automatically caps only the ends that are not snapped to an object.

Control the Appearance of Intersections

You can specify whether DLINE creates a break in a double line, line, or arc when you snap a double line to it. If the Caps option is on, the gap may not be visible.
Snap Double Lines to Objects

You can start or end a double line by snapping to an existing object. To speed up the snap process, you can specify the pixel size of the area where the program searches for object snap candidates. The double line starts or ends by snapping to the object it finds in this search area. Each of the double-line legs adjusts to form a clean junction with the object.

To draw double-line segments

1. Click Draw menu ➤ Double Line.
2. Specify the start point (1).
3. Specify the endpoint (2).
4. Continue specifying points as needed. If you want to remove the previously specified point, enter U or -u at the Command prompt.
5 To connect the start point with the endpoint, enter `cl` (Close). Otherwise, press Enter to complete the double line.

To draw double-line arc segments
1 Click Draw menu ➤ Double Line.
2 Specify a start point.
3 At the Command prompt, enter `a` (Arc).
4 Specify a point if you want to use the three-point method to create the arc, or enter one of the following options at the Command prompt:
   ■ To draw an arc by specifying the center and either the endpoint and/or an included angle, enter `ce`. Then specify a center point, enter an included angle and second point (both optional) and an endpoint.
   ■ To draw an arc by specifying the endpoint and an included angle or center point, enter `e`. Then specify an endpoint and an included angle.
5 If you want to draw a straight line segment, enter `l` (Line) and continue specifying points.
6 To connect the start point with the endpoint, enter `cl` (Close). Otherwise, press Enter to complete the double line.

To set double-line width
1 Click Draw menu ➤ Double Line.
2 At the Command prompt, enter \texttt{w} (Width). Then specify a new width.
3 Specify a point to start drawing the double line.

**To set the dragline for drawing double lines**
1 Click Draw menu $\Rightarrow$ Double Line.
2 At the Command prompt, enter \texttt{d} (Dragline).
3 Specify the offset of the double lines from the points you specify. Enter \texttt{L} (Left), \texttt{c} (Center), or \texttt{r} (Right). If you want to offset the placement point a specific distance from the center line, enter a distance.
4 Specify a point to continue drawing the double line.

**To set the endcaps for double lines**
1 Click Draw menu $\Rightarrow$ Double Line.
2 At the Command prompt, enter \texttt{ca} (Caps).
3 Enter one of the following options:
   - To close both ends of the double line, enter \texttt{b} (Both).
   - To close only the endpoint, enter \texttt{e} (Endpoint).
   - To close only the start point, enter \texttt{s} (Start Point).
   - To leave both ends open, enter \texttt{n} (None).
   - To automatically close all the ends that are not snapped to another object (the default), enter \texttt{a} (Auto).
4 Specify a point to continue drawing the double line.

![Diagram of double lines with different endcaps: both, endpoint, start point, none]

**To create a gap where double lines intersect**
1 Click Draw menu $\Rightarrow$ Double Line.
2 At the Command prompt, enter `b` (Break).
3 Enter `on`.
4 At the Command prompt, enter `s` (Snap).
5 Enter `on`.
6 Specify a point (1) near another double line, line, or arc.

To snap to objects when drawing double lines
1 Click Draw menu ➤ Double Line.
2 At the Command prompt, enter `s` (Snap).
3 Specify one of these options:
   ■ To turn on Snap, enter `on`.
   ■ To change the size of the snap area, enter `s` (Size). Then enter a number between 1 and 10 to specify the number of pixels in the search area.
4 Specify a start point.
5 Specify a point on an existing object to end the double line.

Quick Reference

Commands
DLINE

Creates a double line using straight line segments and arcs.
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.

To draw freehand sketches

1. At the Command prompt, enter **sketch** and press Enter.
Press Enter again to accept the last saved type, increment, and tolerance values.

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.

![Diagram of arc creation](image)

**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.

![Diagram of arc creation](image)

**Draw Arcs by Specifying Start, Center, Angle**

You can create an arc using a start point, center, and an included angle.

The distance between the start point and the center determines the radius. The other end of the arc is determined by specifying an included angle that uses the center of the arc as the vertex. The resulting arc is always created counterclockwise from the start point.

Using different options, you can specify either the start point first or the center point first.
The included angle determines the endpoint of the arc. Use the Start, End, Angle method when you know both endpoints but cannot snap to a center point.

**Draw Arcs by Specifying Start, Center, Length**

You can create an arc using a start point, center, and the length of a chord. The distance between the start point and the center determines the radius. The other end of the arc is determined by specifying the length of a chord between the start point and the endpoint of the arc. The resulting arc is always created counterclockwise from the start point.

Using different options, you can specify either the start point first or the center point first.

The length of the chord of the arc determines the included angle.

**Draw Arcs by Specifying Start, End, Angle**

You can create an arc using a start point, endpoint, and an included angle.
The included angle between the endpoints of the arc determines the center and the radius of the arc.

**Draw Arcs by Specifying Start, End, Direction**

You can create an arc using a start point, endpoint, and a tangent direction at the start point.

The tangent direction can be specified either by locating a point on the desired tangent line, or by entering an angle. You can determine which endpoint controls the tangent by changing the order in which you specify the two endpoints.

**Draw Arcs by Specifying Start, End, Radius**

You can create an arc using a start point, endpoint, and a radius.

The direction of the bulge of the arc is determined by the order in which you specify its endpoints. You can specify the radius either by entering it or by specifying a point at the desired radius distance.

**Draw Contiguous Tangent Arcs and Lines**

Immediately after you create an arc, you can start a line that is tangent to the arc at an endpoint by starting the LINE command and pressing Enter at the Specify First Point prompt. You need to specify only the line length.

Immediately after you create a line or an arc, you can start an arc that is tangent at an endpoint by starting the ARC command and pressing Enter at the Specify Start Point prompt. You need to specify only the endpoint of the new arc.
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 413
- Draw Isometric Circles on page 1385
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.
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.
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.

PLINE
- Creates a 2D polyline.

VIEWRES
- Sets the resolution for objects in the current viewport.

**System Variables**

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

HPBOUND
- Controls the object type created by 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.

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.

See also:
- Draw Isometric Circles on page 1385
- Break and Join Objects on page 606

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).
Specify the center of the circle.
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

See also:
■ Modify Splines on page 615
■ Break and Join Objects on page 606

A spline is a smooth curve that passes through or near a given set of points. The SPLINE command creates a type of curve known as a nonuniform rational B-spline (NURBS). A NURBS curve produces a smooth curve between control vertices or fit points. The spline on the left is drawn with fit points, and the spline on the right is drawn with control vertices.
Close the spline so that the start and endpoints are coincident and tangent.

**Fit Points vs. Control Vertices**

There are different drawing options available depending on whether you use fit points or control vertices.

- **Fit Points (Interpolated)** - Specify the knot parameterization and tolerance settings, but not the degree setting (a degree 3 spline is always created).

- **Control Vertices** - You can specify the tolerance and degree settings, but not the knot parameterization.

**To draw a spline in 2D workspace**

1. Click Home tab ➤ Draw panel ➤ Spline.
2. Click in the drawing area to create the spline.
3. When you are done, press Enter.

**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
PEDIT
Edits polylines.
PLINE
Creates a 2D polyline.
SPLINE
Creates a smooth curve that passes through fit points or near control vertices.
SPLINEDIT
Edits a spline or spline-fit polyline.

System Variables
PLINECONVERTMODE
Specifies the fit method used in converting splines to polylines.

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 (\(\)) 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.

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 two-dimensional enclosed areas that have physical properties such as centroids or centers of mass. You can combine existing regions into a single, complex region to calculate area.

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.

Regions can be used for

■ Applying hatching and shading
■ Analyzing properties, such as area, using MASSPROP
■ Extracting design information, such as the centroid

You can create regions out of multiple loops and out of open curves whose endpoints are connected and form loops. You cannot form regions from open objects that intersect to form a closed area: for example, intersecting arcs or self-intersecting curves.
You can create regions using the REGION and BOUNDARY commands. You create composite regions by combining, subtracting, or finding the intersection of regions.

Objects combined using UNION:

Objects combined using SUBTRACT:

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. 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 REGION or BOUNDARY command. They are removed when you specify a closed boundary, or by using the REDRAW, REGEN, or REGENALL command.

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.
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 adding

1. Click Modify menu ➤ Region ➤ 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.

NOTE You can make a new boundary set to limit the objects used to determine the boundary.
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
MASSPROP
REGION
   Converts an object that encloses an area into a region object.
SUBTRACT
UNION

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. REVCLGOUD 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.

REV CLOUD 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 REV CLOUD before you begin the command. REV CLOUD 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

REV CLOUD

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.
Change Existing Objects

You can select objects, view and edit object properties, and perform general and object-specific editing operations.

Select Objects
You have a wide range of options when you need to select objects for editing operations.

Select Objects Individually
At the Select Objects prompt, you can select one or more objects individually.

Use the Pickbox Cursor
When the square pickbox cursor is in position to select an object, the object is highlighted. Click to select the object.

You can control the size of the pickbox in the Options dialog box, Selection tab.

Select Objects Close Together
It is difficult to select objects that are close together or lie directly on top of one another. The example shows two lines and a circle that lie within the pickbox.
If selection preview is turned on, you can cycle through the objects by rolling over the object on top to highlight it, and pressing and holding Shift and then pressing Spacebar continuously. When the required object is highlighted, left-click to select it.

If selection preview is turned off, hold down Shift + Spacebar and click to cycle through these objects, one after the other, until the one you want is selected. Press Esc to turn off cycling.

**Remove Selection from Objects**

Remove objects from the current selection set by holding down Shift and selecting them again.

**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.

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

PICKAUTO
Controls automatic windowing at the Select Objects prompt.

PICKBOX
Sets the object selection target height, in pixels.

PICKDRAG
Controls the method of drawing a selection window.

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

At the Select Objects prompt, you can select many objects at the same time.

Specify a Rectangular Selection Area

Specify opposite corners to define a rectangular area. The background inside the area changes color and becomes transparent. The direction that you drag your cursor from the first point to the opposite corner determines which objects are selected.

- **Window selection.** Drag your cursor from left to right to select only objects that are entirely enclosed by the rectangular area.

- **Crossing selection.** Drag your cursor from right to left to select objects that the rectangular window encloses or crosses.

With a window selection, usually the entire object must be contained in the rectangular selection area. However, if an object with a noncontinuous (dashed) linetype is only partially visible in the viewport and all the visible vectors of the linetype can be enclosed within the selection window, the entire object is selected.
Specify an Irregularly Shaped Selection Area

Specify points to define an irregularly shaped area. Use window polygon selection to select objects entirely enclosed by the selection area. Use crossing polygon selection to select objects enclosed or crossed by the selection area.

Specify a Selection Fence

In a complex drawing, use a selection fence. A selection fence looks like a polyline and selects only the objects it passes through. The circuit board illustration shows a fence selecting several components.

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 at the Select Objects prompt.

PICKBOX
Sets the object selection target height, in pixels.

PICKDRAG
Controls the method of drawing a selection window.

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

PREVIEWEFFECT
Specifies the visual effect used for previewing selection of objects.

Prevent Objects from Being Selected
You can prevent objects on specified layers from being selected and modified by locking those layers.

Typically, you lock layers to prevent accidental editing of particular objects. Other operations are still possible when a layer is locked. For example, you can make a locked layer current, and you can add objects to it. You can also use inquiry commands (such as LIST), use object snaps to specify points on objects on locked layers, and change the draw order of objects on locked layers.
To help you differentiate between locked and unlocked layers, you can do the following:

- Hover over an object to see whether a lock icon is displayed
- Dim the objects on locked layers

**NOTE** Grips are not displayed on objects that are on locked layers.

**To lock or unlock a layer**

1. Click Home tab ➤ Layers panel ➤ Layer Properties.
2. In the Layer Properties Manager, click the padlock for the layers that you want to lock.
3. Click OK.
   - If the padlock is closed, the layer is locked and objects on that layer cannot be selected.

**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

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 Quick Select or 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 533
- Work with Layers on page 311

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.
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 at the Select Objects prompt.

PICKBOX
Sets the object selection target height, in pixels.

PICKDRAG
Controls the method of drawing a selection window.

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

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 or change the appearance of selection previewing (Options dialog box, Selection tab). When the PICKBOX system variable is set to 0, selection previewing of objects is not available.

**Control the Appearance of Selected Objects**

By default, selected objects are displayed with dashed lines. You can increase program 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**

Options on the Selection tab of the Options dialog box control default selection methods:

- Use selection previewing and selection area effects to preview selection.
- Select objects before entering a command (noun-verb selection) or after entering a command. (PICKFIRST)
- Press Shift to append objects to the selection set. (PICKADD)
- Click and drag to create a selection window. Otherwise you must click twice to define the corners of a selection window. (PICKDRAG)
- Start Window or Crossing selection automatically when you click an empty space. Otherwise, you must enter c or w to specify window crossing selection. (PICKAUTO)
- Change the size of the pickbox. (PICKBOX)
- Select all objects in a group when you select one object in that group.
- Include the boundary in the selection set when you select a hatch.
To change the size of the pickbox cursor

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Selection tab, under Pickbox Size, move the slider until the pickbox is the size you want to use.
3. Click OK.

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 at the Select Objects prompt.
PICKBOX
Sets the object selection target height, in pixels.

PICKDRAG
Controls the method of drawing a selection window.

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

PREVIEWEFFECT
Specifies the visual effect used for previewing selection of objects.

PREVIEWFILTER
Excludes specified object types from selection previewing.

SELECTIONAREA
Controls the display of effects for selection areas.

SELECTIONAREAOPACITY
Controls the transparency of the selection area during window and crossing selection.

SELECTIONPREVIEW
Controls the display of selection previewing.

WINDOWAREACOLOR
Controls the color of the transparent selection area during window selection.

Group Objects
A group is a saved set of objects that you can select and edit together or separately as needed. Groups provide an easy way to combine drawing elements that you need to manipulate as a unit.

Overview of Groups
A group is a saved set of objects that you can select and edit together or separately as needed. Groups provide an easy way to combine drawing elements that you need to manipulate as a unit. You can create them quickly and with
a default name, or you can use the Group Manager to assign a name from the start.

You can change the components of groups as you work by adding or removing objects.

In some ways, groups resemble blocks, which provide another method of combining objects into a named set. For example, the groups you create are saved from session to session. However, you can edit individual objects in groups more easily than you can edit them in blocks, which must be exploded first. Unlike blocks, groups cannot be shared with other drawings.

Quick Reference

GROUP

Creates and manages saved sets of objects called groups.

QKUNGROUP

Removes a group definition from a drawing.

PICKSTYLE

Controls the use of group selection and associative hatch selection.

Create Groups

In addition to choosing the objects that will become the members of a group, you can give the group a name and description.

When you create a group, you can give the group a name and description. If you copy a group, the copy is given the default name Ax and is considered unnamed. Unnamed groups are not listed in the Object Grouping dialog box unless you select Include Unnamed.

If you choose a member of a group that can be selected for inclusion in a new group, all members of the former group are included in the new group.

There are two ways to create groups:

■ **Unnamed groups**. To create an unnamed group, first select the objects you want to group. Click Group on the Group toolbar. A default name such as *A1 or *A2 is assigned. You can use the Group Manager later to assign a name and description or change the components in this unnamed group.
**Named groups.** Create a named group in the Group Manager. Select the objects you want to group and then select Create Groups. With this method, you can assign the group a name and description as you create it.

The objects in your drawing can be members of more than one group, and groups themselves can be nested in other groups. You can ungroup a nested group to restore the original group configuration.

Named groups are not maintained when you use a drawing as an external reference or insert it as a block. However, you can bind and then explode the external reference or explode the block to make the group available as an unnamed group.

**NOTE** Avoid creating large groups containing hundreds or thousands of objects. A large group significantly degrades the performance of this program.

To create an unnamed group

1. Select the objects to associate into a group.
2. Click Tools menu ➤ 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 Group Manager unless Include Unnamed Groups is selected.

To create a named group

1. Click Tools menu ➤ Group Manager.
2. Select the objects to associate into a group.
3. In the Group Manager, click the Create Group button.
4. Under Group, enter a group name and press Enter.
   The selected objects are grouped together into a named group.
5. Click in the description area and enter a description of the group (optional).
Quick Reference

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 then can edit the group as a unit. Selecting an object that belongs to multiple groups selects all the groups. You can turn off group selection to select objects individually.

Select a Group

The easiest way to select a group is to click one of its objects with the pointing device. However, if you need to access a nested group or find and select a specific group, use the Group Manager. You also can select groups by name by entering `group` and the group name at any Select Objects prompt.

Select Individual Objects in a Group

To modify objects in a group individually, turn off group selection using one of the following methods:

- The Group Selection On/Off button on the Group toolbar toggles group selection for all groups in your drawing. All objects, including those in nested groups, become available for editing. Choosing this button sets the PICKSTYLE system variable: When PICKSTYLE is on (1), groups are selectable.

- The light bulb icon in the Selectable column in the Group Manager toggles group selection for the selected group.

NOTE When PICKSTYLE is off (0), no groups are selectable, even if the light bulb in the Group Manager is yellow.
To turn group selection on and off for all groups

- On the Group toolbar, click Group Selection On/Off.

The status of the PICKSTYLE system variable is displayed at the Command prompt. When PICKSTYLE is set to 1 or 3, all groups are selectable.

To control selectability for a specific group

1. Click Tools menu ➤ Group Manager.
2. In the Group Manager, select a group.
3. Under Selectable, click the light bulb to switch between gray and yellow.

**NOTE** If the PICKSTYLE system variable is off (0 or 2), no groups are selectable regardless of the color of the light bulb.

To select groups by name

1. Click Tools menu ➤ Group Manager.
2. In the Group Manager, under Group, select the names of the groups you want to select.
3. Click the Select Group button.
   All of the objects in the selected groups are highlighted with grips.

**NOTE** To specify a group at the Command prompt or at a Select Objects prompt, enter `group` followed by the name of the group.

To identify which groups an object belongs to

1. In the drawing area, select an object.
2. Click Tools menu ➤ Group Manager.
3. With the cursor over any group name, right-click. Click Clear All.
4. With the cursor over any group name, right-click. Click Show Groups for Selected Objects.
   The objects are highlighted in the Group column.
Quick Reference

GROUP
Creates and manages saved sets of objects called groups.

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 542.

Change Group Components, Name, or Description
With the Group Manager, you can specify objects to be added to or removed from a group at any time. You can also revise a group's name or description. If deleting an object or removing it from a group leaves the group empty, the group remains defined but without any members.

NOTE Exploding an object such as a block instance or hatch that belongs to a group does not automatically add the resulting components to any group.

Remove Groups
You can delete a group definition by using the Ungroup options on the menu or toolbar in the Group Manager.

As a result, the group is disbanded but the members are not changed in any other way.

To remove a group
1. Select the group you want to remove.
2. Click Tools menu ➤ Ungroup.
To add objects to a group
1. Click Tools menu ➤ Group Manager.
2. In the Group Manager, select the group name from the list of groups.
3. Select the objects to add to the group.
4. Click Add to Group.
   The objects are added to the selected group.

To remove objects from a group
1. Click Tools menu ➤ Group Manager.
2. In the Group Manager, under Group, select the group you want to change.
3. To turn off individual selection for that group, under Selectable, click the light bulb to make it gray.
4. Select the objects you want to remove from the group.
5. In the Group Manager, click Remove from Group.
   The objects are removed from the selected group.

Quick Reference
GROUP
   Creates and manages saved sets of objects called groups.
QKUNGROUP
   Removes a group definition from a drawing.
PICKSTYLE
   Controls the use of group selection and associative hatch selection.

Correct Mistakes
You can backtrack your recent actions using one of several methods.

Undo a Single Action
The simplest method of backtracking is to use Undo on the Standard toolbar or the U command to undo a single action. Many commands include their
own U (undo) option so that you can correct mistakes without leaving the command. When you are creating lines and polylines, for example, enter `u` to undo the last segment.

**NOTE** By default, the UNDO command is set to combine consecutive pan and zoom commands into a single operation when you undo or redo. However, pan and zoom commands that are started from the menu are not combined, and always remain separate actions.

**Undo Several Actions at Once**

Use the Mark option of UNDO to mark an action as you work. You can then use the Back option of UNDO to undo all actions that occurred after the marked action. Use the Begin and End options of UNDO to define a set of actions to be treated as a group.

You can also undo several actions at once with the Undo list on the Standard toolbar.

**Reverse the Effect of Undo**

You can reverse the effect of a single U or UNDO command by using REDO immediately after using U or UNDO.

You can also redo several actions at once with the Redo list on the Standard toolbar.

**Erase Objects**

You can erase any object that you draw. If you accidentally erase the wrong object, you can use the UNDO command or the OOPS command to restore it.

For more information, see Erase Objects on page 548.

**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.

Clean Up the Display
You can remove the plus-shaped markers called blips and stray pixels that may be left over from some editing operations from the display area.

■ To remove blips, use REDRAW.
■ To remove stray pixels, use REGEN.

See also:
■ Correct Mistakes on page 545

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 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 remove plus-shaped marker blips

Click View menu ➤ Redraw.

To purge an unused linetype

1 Click Tools tab ➤ Drawing Utilities panel ➤ Purge.

   The Purge dialog box displays a tree view of object types with items that can be purged.

2 To purge unreferenced linetypes, use one of the following methods:
   - To purge all unreferenced linetypes, select Linetypes.
   - To purge specific linetypes, double-click Linetypes to expand the tree view. Then select the linetypes to be purged.

   If the item you want to purge is not listed, select View Items You Cannot Purge.
3 You are prompted to confirm each item in the list. If you do not want to confirm each purge, clear the Confirm Each Item to Be Purged option.

4 Click Purge.
   To confirm the purging of each item, respond to the prompt by choosing Yes or No, or Yes to All if more than one item is selected.

5 Click Close.

To remove zero-length geometry and empty text objects

1 Click Tools tab ➤ Drawing Utilities panel ➤ Purge. The Purge dialog box displays.

2 Select Purge zero-length geometry and empty text objects.

3 Click Purge.

4 Click Close.

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.

PURGE
   Removes unused items, such as block definitions and layers, from the drawing.

REDRAW
   Refreshes the display in the current viewport.

REGEN
   Regenerates the entire drawing from the current viewport.
UNDO

Reverses the effect of commands.

Use Windows Cut, Copy, and Paste

When you want to use objects from a drawing file in another application, you can cut or copy these objects to the Clipboard and then paste them from the Clipboard into the other application.

Cut Objects

Cutting deletes selected objects from the drawing and stores them on the Clipboard. The objects are now available to be pasted into other Microsoft® Windows® documents.

Copy Objects

You can use the Clipboard to copy part or all of a drawing into a document created by another application. The objects are copied in vector format, which retains the high resolution in other applications. These objects are stored in WMF (Windows metafile) format in the Clipboard. The information stored in the Clipboard can then be embedded in the other document. Updating the original drawing does not update the copy embedded in the other application.

Paste Objects

Applications use different internal formats to store Clipboard information. When you copy objects to the Clipboard, information is stored in all available formats. When you paste the Clipboard contents into a drawing, the format that retains the most information is used. However, you can override this setting and convert pasted information to AutoCAD LT format.

Because it is the easiest format to edit, the AutoCAD LT format is the preferred format for copying objects to and from AutoCAD LT. 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 LT are of higher resolution than bitmapped images (BMP files) but are not as easily manipulated as AutoCAD LT 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 LT format, the object is inserted as a block reference. To edit the pasted information, explode the block reference into its component objects. When you convert a Windows metafile stored on the Clipboard to AutoCAD LT 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 LT 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.

CLIPBOARD
Indicates the status of the Clipboard.

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 1308
- Modify Existing Dimensions on page 1023

Choose a Method to Modify Objects
Access object editing options using the following methods:

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<th>Descriptions</th>
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<td>Enter a command and then select the objects to modify. Alternatively, select the objects first and then enter a command.</td>
</tr>
<tr>
<td>Shortcut menu</td>
<td>Select and right-click an object to display a shortcut menu with relevant editing options.</td>
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<tr>
<td>Methods</td>
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<td>---------------</td>
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<tr>
<td>Double-click</td>
<td>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.)</td>
</tr>
<tr>
<td>Grips</td>
<td>Control grip behavior with the following methods:</td>
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<tr>
<td></td>
<td>■ <strong>Grip Modes.</strong> Click a grip and right-click to select one of the Grip modes (Stretch, Move, Rotate, Scale, or Mirror).</td>
</tr>
<tr>
<td></td>
<td>■ <strong>Multi-functional grip-editing options.</strong> Select a polyline, spline, or non-associative polyline hatch object and hover over a grip to access additional options for reshaping the object.</td>
</tr>
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See also:
- Change Text on page 915
- Select Objects on page 521
- Modify Existing Dimensions on page 1023
- Display and Change the Properties of Objects on page 307
- Work with Custom and Proxy Objects on page 1308
- “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.

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.

Edit Objects with Grips
 Grips are displayed at strategic points on selected objects. Manipulate grips to modify objects.

Use Grip Modes
 Grip modes control how an object behaves when manipulated with grips. Use Grip modes to stretch, move, scale, rotate, or mirror an object.

To use a Grip mode, select a grip (base grip) to act as the base point for the action. (A selected grip is also called a hot grip.) Then press Enter or Spacebar to cycle through the Grip modes. You can also use shortcut keys or right-click to see all of the modes.
NOTE Grips are not displayed on objects that are on locked layers.

**Stretch with Grips**

You can stretch an object by moving selected grips to new locations. Grips on text, block references, midpoints of lines, centers of circles, and point objects move the object rather than stretching it. This is an excellent method for moving block references and adjusting dimensions.

**NOTE** When a 2D object lies on a plane other than the current UCS, the object is stretched on the plane on which it was created, not on the plane of the current UCS.

**Move with Grips**

You can move objects by the grip selected. Selected objects are highlighted and are moved the direction and distance of the next point location you specify.

**Scale with Grips**

You can scale selected objects relative to a base point. Increase the size of an object by dragging outward from the base grip and specifying a point location, or decrease the size by dragging inward. Alternatively, you can enter a value for relative scaling.

**Mirror with Grips**

You can mirror selected objects across a temporary mirror line. Turning Ortho on helps you specify a vertical or horizontal mirror line.
Rotate with Grips
You can rotate selected objects around a base point by dragging and specifying a point location. Alternatively, you can enter an angle value. This is an excellent method for rotating block references.

Select and Modify Multiple Grips
You can use more than one grip as the base grips for the action. When you select more than one grip (also called *multiple hot grip selection*), the shape of the object is kept intact between the selected grips. To select more than one grip, press and hold the Shift key, and then select the appropriate grips.

Limit the Display of Grips 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.

NOTE Grips are not displayed on objects that are on locked layers.

Work with Quadrant Grips
For quadrant grips on circles and ellipses, distance is measured from the center point, not the selected grip. For example, in Stretch mode, you can select a quadrant grip to stretch a circle and then specify a distance at the Command prompt for the new radius. The distance is measured from the center of the circle, not the selected quadrant. If you select the center point to stretch the circle, the circle moves.

See also:
- Choose a Method to Modify Objects on page 554
- Modify Polylines on page 609
- Modify Hatches and Solid-Filled Areas
- Use Dynamic Input on page 406
- Work with Dynamic Blocks in Drawings on page 662
To turn on grips
1 At the Command prompt, enter options.
2 In the Options dialog box, Selection tab, select Enable Grips.
3 Click OK.

To stretch an object using grips
1 Select the object to stretch.
2 Select a base grip on the object.
The selected grip is highlighted, and Stretch, the default grip mode, is active.
3 Move the pointing device and click.
The selected object is stretched as the grip moves.

NOTE To copy the selected object while stretching, press and hold the Ctrl key.

To stretch using more than one grip
1 Select several objects to stretch.
2 Hold down Shift and click several grips so that they are highlighted.
3 Release Shift and select a grip as the base grip by clicking the grip.
The default grip mode, Stretch, is active.
4 Move the pointing device and click.
The selected grips act in unison and the selected objects are stretched.

To move objects using grips
1 Select the objects to move.
2 Select a base grip on an object by clicking the grip.
The selected grip is highlighted, and Stretch, the default grip mode, is active.
3 Cycle through the grip modes by pressing Enter until the grip mode Move appears.
   Alternatively, you can right-click to display a shortcut menu of modes and options.
4 Move the pointing device and click. The selected objects are moved along with the grip.

**NOTE** To copy the selected object while moving, press and hold the Ctrl key.

**To rotate objects using grips**

1 Select the objects to rotate.

2 Select a base grip on an object by clicking the grip. The selected grip is highlighted, and Stretch, the default grip mode, is active.

3 Cycle through the grip modes by pressing Enter until the grip mode Rotate appears. Alternatively, you can right-click to display shortcut menu modes and options.

4 Move the pointing device and click. The selected objects are rotated around the base grip.

**NOTE** To copy the selected object while rotating, press and hold the Ctrl key.

**To scale objects using grips**

1 Select the objects to scale.

2 Select a base grip on an object by clicking the grip. The selected grip is highlighted, and Stretch, the default grip mode, is active.

3 Cycle through the grip modes by pressing Enter until the grip mode Scale appears. Alternatively, you can right-click to display shortcut menu modes and options.

4 Enter the scale factor or drag and click to specify a new scale.

**NOTE** To copy the selected object while scaling, press and hold the Ctrl key.

**To mirror objects using grips**

1 Select the objects to mirror.
2 Select a base grip on an object by clicking the grip. The selected grip is highlighted, and the default grip mode, Stretch, is active.

3 Cycle through the grip modes by pressing Enter until the grip mode Mirror appears. Alternatively, you can right-click to display shortcut menu modes and options.

4 Click to specify the second point of the mirror line. Turning on Ortho mode is often useful when mirroring objects.

Quick Reference

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.

GRIPOVER
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.
GRIP TIPS

Controls the display of grip tips and Ctrl-cycling tooltips.

Modify Objects with Multi-Functional Grips

Modify polylines, splines, and non-associative polyline hatch objects with multi-functional grips.

Control the display of these grips with the GRIPS system variable.

With multi-functional grips, you can

- **Modify the position, size, and orientation of objects.** Use Grip modes on page 556 to move, rotate, scale, or mirror objects.
- **Reshape Objects.** Use the multi-functional grip-editing options to edit vertices, fit points, control points, segment types, and tangent directions.

To activate a multi-functional grip, select the grip, or hover over it and choose a multi-functional grip-editing option from the dynamic menu. Once a grip is active, change the grip behavior with the Hot Grip shortcut menu, or by cycling through options as follows:

- Press Enter or Spacebar to cycle through the Grip modes
- Press Ctrl to cycle through the multi-functional grip-editing options.

**NOTE** If the object is constrained, the first Ctrl relaxes constraints.

Control the access methods to multi-functional grips with the GRIMPULIFUNCTIONAL system variable.

**To reshape an object with multi-functional grips**

1. Select a polyline, spline, or non-associative polyline hatch object.
2. Hover the cursor over a multi-functional grip.
3. From the dynamic menu, click a multi-functional grip-editing option.
4. While manipulating the grip, press Ctrl to cycle through the other multi-functional grip-editing options.
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.

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.

GRIPTIPS
Controls the display of grip tips and Ctrl-cycling tooltips.

GRIPMULTIFUNCTIONAL
Specifies the access methods to multi-functional grips.

Make Multiple Copies with Grips
You can create multiple copies of objects as you modify them with any of the grip modes.
For example, by using the Copy option, you can rotate the selected objects, leaving copies at each location you specify with the pointing device.

You can also make multiple copies by holding down Ctrl as you select the first point. For example, with the Stretch grip mode, you can stretch an object, such as a line, and then copy it to any point in the drawing area. Multiple copies continue being made until you turn off grips.

**NOTE** When you use grips to make multiple copies of an object that contains multiple , only the current scale representation is copied.

### Define an Offset Snap or a Rotation Snap

You can place multiple copies at regularly spaced intervals with an offset snap. The offset snap is defined by the distance between an object and the next copy. In the lighting layout below, the first copy of the light fixture symbol is placed at an offset of two units. All subsequent copies are then placed two units apart.

If you hold down Ctrl while you select multiple copy points with the pointing device, the graphics cursor snaps to an offset point based on the last two points you selected. In the illustration below, the midpoint of line 1 is at coordinate 8,5. Based on that midpoint, line 2 was copied using the Ctrl key and Stretch grip mode; its midpoint is at 9,5. The third line snaps to an offset based on the coordinate values 10,5.
Similarly, you can place multiple copies at angular intervals around a base grip with a rotation snap. The rotation snap is defined as the angle between an object and the next copy when you are using Rotate grip mode. Hold down Ctrl to use the rotation snap.

To create copies in any grip mode

1. Select the objects to copy.
2. Select a base grip on an object by clicking the grip. The selected grip is highlighted, and the default grip mode, Stretch, is active.
3. Cycle through the grip modes by pressing Enter until the grip mode you want appears. Alternatively, you can right-click to display shortcut menu modes and options.
4. Enter c (Copy) or press and hold the Ctrl key while you stretch, move, rotate, or scale. Copies continue being made until you turn off grips.
5. Enter or specify the additional input required for the current grip mode.
6. Turn off grips by pressing Enter, Spacebar, or Esc.
To create an offset snap for multiple copies using grips

1. Select the objects to copy.
2. Select a base grip on an object by clicking the grip.
   The selected grip is highlighted, and the default grip mode, Stretch, is active.
3. Cycle through the grip modes by pressing Enter until the grip mode Move appears.
   Alternatively, you can right-click to display shortcut menu modes and options.
4. Enter c (Copy).
5. Move the cursor and click.
   The offset snap is the distance between the grip you selected and the location you specified for the copy.
6. Hold down Ctrl and place additional copies by specifying additional locations.
   These copies are created at the same offset snap distance as the last copy.
7. Turn off grips by pressing Enter, Spacebar, or Esc.

To create a rotation snap for multiple rotated copies using grips

1. Select the objects to rotate.
2. Select a base grip on an object by clicking the grip.
   The selected grip is highlighted, and the default grip mode, Stretch, is active.
3. Cycle through the grip modes by pressing Enter until the grip mode Rotate appears.
   Alternatively, you can right-click to display shortcut menu modes and options.
4. Enter c (Copy).
5. Move the pointing device and click.
   The rotation snap is the angle between the grip you selected and the location you specified for the copy.
6. Hold down Ctrl and place additional copies by specifying additional locations.
These copies are created at the same rotation snap angle as the first copy.

7 Turn off grips by pressing Enter, Spacebar, or Esc.

To mirror objects and retain the originals using grips
1 Select the objects to mirror.
2 Select a base grip on an object by clicking the grip.
   The selected grip is highlighted, and the default grip mode, Stretch, is active.
3 Cycle through the grip modes by pressing Enter until the grip mode Mirror appears.
   Alternatively, you can right-click to display shortcut menu modes and options.
4 Hold down Ctrl (or enter c for Copy) to retain the original image, and specify the second point of the mirror line.
   Turning on Ortho mode is often useful when mirroring objects.
5 Turn off grips by pressing Enter, Spacebar, or Esc.

Quick Reference

OPTIONS
   Customizes the program settings.
GRIPBLOCK
   Controls the display of grips in blocks.
GRIPCOLOR
   Controls the color of unselected grips.
GRIPCOURT
   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.

![Control Grips in Blocks](image)

**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. Click Home tab ➤ Modify panel ➤ Move. Then select the object to be moved (1). Specify the base point for the move (2) followed by a second point (3). The object is moved the distance and direction of point 2 to point 3.
Specify Distance with Relative Coordinates

You can move an object using a relative distance by entering coordinate values for the first point and pressing Enter for the second point. The coordinate values are used as a relative displacement rather than the location of a base point.

NOTE Do not include an @ sign as you normally would for relative coordinates, because relative coordinates are expected.

To copy objects a specified distance, you can also use direct distance entry with Ortho mode and polar tracking. For more information, see Enter Direct Distances on page 440.

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.

Use Alternate Methods

You can use grips to move and copy objects quickly. See Edit Objects with Grips on page 556.

You can also select objects and drag them to a new location; press Ctrl to make a copy. Using this method, you can drag objects between open drawings and other applications. If you drag with the right mouse button instead of the left, a shortcut menu is displayed. The menu options include Move Here, Copy Here, Paste as Block, and Cancel. See Embed OLE Objects in Drawings on page 1226.
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

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.

Direct Distance Entry (Command Modifier)
    Locates the next point at a specified distance in the direction of your cursor.
Rotate Objects

You can rotate objects in your drawing around a specified base point.

To determine the angle of rotation, you can enter an angle value, drag using the cursor, or specify a reference angle to align to an absolute angle.

Rotate an Object by a Specified Angle

Enter a rotation angle value from 0 to 360 degrees. You can also enter values in radians, grads, or surveyor bearings. Entering a positive angle value rotates the objects counterclockwise or clockwise, depending on the Direction Control setting in the Drawing Units dialog box.

Rotate an Object by Dragging

Drag the object around the base point and specify a second point. Use Ortho mode, polar tracking, or object snaps for greater precision.

For example, you can rotate the plan view of a house by selecting the objects (1), specifying a base point (2), and specifying an angle of rotation by dragging to another point (3).

Rotate an Object to an Absolute Angle

With the Reference option, you can rotate an object to align it to an absolute angle.

For example, to rotate the part in the illustration so the diagonal edge rotates to 90 degrees, you select the objects to be rotated (1, 2), specify the base point (3), and enter the Reference option. For the reference angle, specify the two endpoints of the diagonal line (4, 5). For the new angle, enter 90.
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.
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.

Quick Reference

**ROTATE**
Rotates objects around a base point.

**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.

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.
Quick Reference

ALIGN
Aligns objects with other objects in 2D and 3D.

Copy, Offset, or Mirror Objects
You can create duplicates of objects in your drawing that are either identical or similar to selected objects.

Copy Objects
You can create duplicates of objects at a specified distance and direction from the originals.

Use coordinates, grid snap, object snaps, and other tools to copy objects with precision.

You can also use grips to move and copy objects quickly. See Edit Objects with Grips on page 556.

Specify Distance with Two Points
Copy an object using the distance and direction specified by a base point followed by a second point. In this example, you copy the block representing an electronic component. Click Edit menu ➤ Copy. Then select the original object to be copied. Specify the base point for the move (1) followed by a second point (2). The object is copied the distance and direction of point 1 to point 2.
Specify Distance with Relative Coordinates

Copy an object using a relative distance by entering coordinate values for the first point and pressing Enter for the second point. The coordinate values are used as a relative displacement rather than the location of a base point.

**NOTE** Do not include an @ sign as you normally would for relative coordinates, because relative coordinates are expected.

To copy objects a specified distance, you can also use direct distance entry with Ortho mode and polar tracking. For more information, see Enter Direct Distances on page 440.

Create Multiple Copies

The COPY command repeats automatically by default. To exit the command, press Enter. To change the default, use the COPYMODE system variable.

Move and Copy Objects by Dragging

You can also select objects and drag them to a new location using the left mouse button over one of the selected objects; press Ctrl to make a copy. Using this method, you can drag objects between open drawings and other applications.

If you drag with the right mouse button instead of the left, a shortcut menu is displayed after you drag the objects. The menu options include Move Here, Copy Here, Paste as Block, and Cancel.

For information about using object linking and embedding, see Embed OLE Objects in Drawings on page 1226.

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

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.

COPYMODE
Controls whether the COPY command repeats automatically.

Create an Array of Objects
You can create copies of objects in a rectangular or polar (circular) pattern called an array.

For rectangular arrays, you control the number of rows and columns and the distance between each. For polar arrays, you control the number of copies of the object and whether the copies are rotated. To create many regularly spaced objects, arraying is faster than copying.
Create Rectangular Arrays

A rectangular array is built along a baseline defined by the current snap rotation angle. This angle is zero by default, so the rows and columns of a rectangular array are orthogonal with respect to the $X$ and $Y$ axes. The default angle 0 direction setting can be changed in UNITS.

Create Polar Arrays

When you create a polar array, the array is drawn counterclockwise or clockwise, depending on whether you enter a positive or a negative value for the angle to fill.

The radius of the array is determined by the distance from the specified center point to a reference or base point on the last selected object. You can use the default reference point (usually an arbitrary point that coincides with a snap point), or you can specify a new base point to be used as the reference point.

Array in 3D

Limit the Size of Arrays

If you specify a very large number of rows and columns for an array, it may take a long time to create the copies. By default, the number of array elements that can be generated by one command is limited to approximately 100,000. This limit is controlled by the MaxArray setting in the registry.

You can change this limit by setting the MaxArray system registry variable using SETENV and entering a number between 100 and 10000000 (ten million).
NOTE When changing the value of MaxArray, you must enter MaxArray with the capitalization shown.

To create a rectangular array

1. Click Home tab ➤ Modify panel ➤ Array.
2. In the Array dialog box, select Rectangular Array.
3. Click Select Objects.
   The Array dialog box closes. You are prompted for object selection.
4. Select the objects to be arrayed and press Enter.
5. In the Rows and Columns boxes, enter the number of rows and columns in the array.
6. Specify the horizontal and vertical spacing (offsets) between objects by using one of the following methods:
   ■ In the Row Offset and Column Offset boxes, enter the distance between rows and between columns. Adding a plus sign (+) or a minus sign (-) determines direction.
   ■ Click the Pick Both Offsets button to use the pointing device to specify the diagonal corners of a cell in the array. The cell determines the vertical and horizontal spacing of the rows and columns.
   ■ Click the Pick Row Offset or Pick Column Offset button to use the pointing device to specify the horizontal and vertical spacing.
   The example box displays the result.
7. To change the rotation angle of the array, enter the new angle next to Angle of Array.
8. The default angle 0 direction setting can also be changed in UNITS.
9. Click OK to create the array.

To create a polar array

1. Click Home tab ➤ Modify panel ➤ Array.
2. In the Array dialog box, select Polar Array.
3 Next to Center Point, do one of the following:
   ■ Enter an X value and a Y value for the center point of the polar array.
   ■ Click the Pick Center Point button. The Array dialog box closes and you are prompted for object selection. Use the pointing device to specify the center point of the polar array.

4 Click Select Objects.
   The Array dialog box closes and you are prompted for object selection.

5 Select the objects to be arrayed.

6 In the Method box, select one of the following methods:
   ■ Total Number of Items & Angle to Fill
   ■ Total Number of Items & Angle Between Items
   ■ Angle to Fill & Angle Between Items

7 Enter the number of items (including the original object), if available.

8 Use one of the following methods:
   ■ Enter the angle to fill and angle between items, if available. Angle to Fill specifies the distance to fill around the circumference of the array. Angle Between Items specifies the distance between each item.
   ■ Click the Pick Angle to Fill button and the Pick Angle Between Items button. Use the pointing device to specify the angle to fill and the angle between items.
   The example box displays the result.

9 You can set any of the following options:
   ■ To rotate the objects as they are arrayed, select Rotate Items As Copied. The example area displays the result.
   ■ To specify the X,Y base point, select More, clear the Set to Object’s Default option and enter values in the X and Y boxes, or click the Pick Base Point button and use the pointing device to specify the point.

10 Click OK to create the array.
Quick Reference

ARRAY
Creates multiple copies of objects in a pattern.

DSETTINGS
Sets grid and snap, polar and object snap tracking, object snap modes, Dynamic Input, and Quick Properties.

UCS
Manages user coordinate systems.

UNITS
Controls coordinate and angle display formats and precision.

GETENV
Shows values of specified system registry variables.

SETENV
Sets values of specified registry variables.

ANGBASE
Sets the base angle to 0 with respect to the current UCS.

ANGDIR
Sets the direction of positive angles.

SNAPANG
Sets the snap and grid rotation angle for the current viewport relative to the current UCS.

Offset an Object

Offset an object to create a new object whose shape parallels the shape of the original object.

OFFSET creates a new object whose shape parallels the shape of a selected object. Offsetting a circle or an arc creates a larger or smaller circle or arc, depending on which side you specify for the offset.
A highly effective drawing technique is to offset objects and then trim or extend their ends.

You can offset
- Lines
- Arcs
- Circles
- Ellipses and elliptical arcs (resulting in an oval-shaped spline)
- 2D polylines
- Construction lines (xlines) and rays
- Splines

**Special Cases for Offset Polylines and Splines**

2D polylines and splines are trimmed automatically when the offset distance is larger than can otherwise be accommodated.
To offset an object by specifying a distance
1  Click Home tab ➤ Modify panel ➤ Offset.
2  Specify the offset distance.
   You can enter a value or use the pointing device.
3  Select the object to offset.
4  Specify a point on the side where you want to place the new objects.
5  Select another object to offset, or press Enter to end the command.

To offset an object through a point
1  Click Home tab ➤ Modify panel ➤ Offset.
2  Enter t (Through).
3  Select the object to offset.
4  Specify the through point.
5  Select another object to offset, or press Enter to end the command.

Quick Reference
OFFSET
   Creates concentric circles, parallel lines, and parallel curves.
OFFSETDIST
   Sets the default offset distance.

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 MIRRTEXT setting.

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.

**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.
Quick Reference

MIRROR
Creates a mirrored copy of selected objects.

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.

The following example uses the fence selection method to select a series of objects for trimming.
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.

![Diagram of trimming objects](image)

You can extend objects without leaving the TRIM command. Hold down Shift and select the objects to be extended.

**Extend Objects**

Extending operates the same way as trimming. You can extend objects so they end precisely at boundary edges defined by other objects. In this example, you extend the lines precisely to a circle, which is the boundary edge.

![Diagram of extending objects](image)

You can trim objects without leaving the EXTEND command. Hold down Shift and select the objects to be trimmed.

**Trim and Extend Wide Polylines**

2D wide polylines trim and extend at their centerlines. The ends of wide polylines are always square. Trimming a wide polyline at an angle causes portions of the end to extend beyond the cutting edge.

If you trim or extend a tapered 2D polyline segment, the width of the extended end is corrected to continue the original taper to the new endpoint. If this correction gives the segment a negative ending width, the ending width is forced to 0.
Trim and Extend Spline-Fit Polylines

Trimming a spline-fit polyline removes the curve-fit information and changes the spline-fit segments into ordinary polyline segments.

Extending a spline-fit polyline adds a new vertex to the control frame for the polyline.

Trim or Extend in 3D

You can trim or extend an object to any other object in 3D space, regardless of whether the objects are on the same plane or parallel to the cutting or boundary edges. In the TRIM and EXTEND commands, use the Project and Edge options to select one of three projections for trimming or extending:

- The XY plane of the current UCS
- The plane of the current view
- True 3D, which is not a projection

See also:

- Break and Join Objects on page 606

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

**BREAK**

Breaks the selected object between two points.

**EXTEND**

Extends objects to meet the edges of other objects.

**JOIN**

Joins similar objects to form a single, unbroken object.

**LENGTHEN**

Changes the length of objects and the included angle of arcs.
PROPERTIES
Controls properties of existing objects.

TRIM
Trims objects to meet the edges of other objects.

EDGEMODE
Controls how the TRIM and EXTEND commands determine cutting and boundary edges.

PROJMODE
Sets the current Projection mode for trimming or extending.

Resize or Reshape Objects
You can resize objects to make them longer or shorter in only one direction or to make them proportionally larger or smaller.

You can also stretch certain objects by moving an endpoint, vertex, or control point.

Lengthen Objects
With LENGTHEN, you can change the included angle of arcs and the length of the following objects:

- Lines
- Arcs
- Open polylines
- Elliptical arcs
- Open splines.

The results are similar to extending and trimming. You can

- Drag an object endpoint dynamically
- Specify a new length or angle as a percentage of the total length or angle
- Specify an incremental length or angle measured from an endpoint
- Specify the object's total absolute length or included angle
Stretch Objects

With STRETCH, you relocate the endpoints of objects that lie across or within a crossing selection window.

- Objects that are partially enclosed by a crossing window are stretched.
- Objects that are completely enclosed within the crossing window, or that are selected individually, are moved rather than stretched.

To stretch an object, you specify a base point and then a point of displacement.

To stretch with precision, use object snaps, grid snaps, and relative coordinate entry.

Scale Objects Using a Scale Factor

With SCALE, you can make an object uniformly larger or smaller. To scale an object, you specify a base point and a scale factor. Alternatively, you can specify a length to be used as a scale factor based on the current drawing units.

A scale factor greater than 1 enlarges the object. A scale factor between 0 and 1 shrinks the object.

Scaling changes the size of all dimensions of the selected object. A scale factor greater than 1 enlarges the object. A scale factor less than 1 shrinks the object.

NOTE When you use the SCALE command with objects, the position or location of the object is scaled relative to the base point of the scale operation, but the size of the object is not changed.
### Scale Objects Using a Reference Distance

You can also scale by reference. Scaling by reference uses an existing distance as a basis for the new size. To scale by reference, specify the current distance and then the new desired size. For example, if one side of an object is 4.8 units long and you want to expand it to 7.5 units, use 4.8 as the reference length.

You can use the Reference option to scale an entire drawing. For example, use this option when the original drawing units need to be changed. Select all objects in the drawing. Then use Reference to select two points and specify the intended distance. All the objects in the drawing are scaled accordingly.

See also:
- [Break and Join Objects](#) on page 606

### 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

JOIN
   Joins similar objects to form a single, unbroken object.

LENGTHEN
   Changes the length of objects and the included angle of arcs.

PEDIT
   Edits polylines.

PROPERTIES
   Controls properties of existing objects.

SCALE
   Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling.
SPLINEDIT
Edits a spline or spline-fit polyline.

STRETCH
Stretches objects crossed by a selection window or polygon.

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
FILLET can be used to round all corners on a polyline using a single command.

**NOTE** Filleting a hatch boundary that was defined from line segments removes hatch associativity. If the hatch boundary was defined from a polyline, associativity is maintained.

If both objects being filleted are on the same layer, the fillet arc is created on that layer. Otherwise, the fillet arc is created on the current layer. The layer affects object properties including color and linetype.

Use the Multiple option to fillet more than one set of objects without leaving the command.

**Set the Fillet Radius**

The fillet radius is the radius of the arc that connects filleted objects. Changing the fillet radius affects subsequent fillets. If you set the fillet radius to 0, filleted objects are trimmed or extended until they intersect, but no arc is created.

You can hold down Shift while selecting the objects to override the current fillet radius with a value of 0.

**Trim and Extend Filleted Objects**

You can use the Trim option to specify whether the selected objects are trimmed or extended to the endpoints of the resulting arc or left unchanged.
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 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.

Fillet an Entire Polyline

You can fillet an entire polyline or remove fillets from an entire polyline.

If you set a nonzero fillet radius, FILLET inserts fillet arcs at the vertex of each polyline segment that is long enough to accommodate the fillet radius.
If two polyline line segments converge as they approach an arc segment that separates them, FILLET removes the arc segment and replaces it with a fillet arc.

If you set the fillet radius to 0, no fillet arcs are inserted. If two polyline line segments are separated by one arc segment, FILLET removes that arc and extends the lines until they intersect.

**Fillet Parallel Lines**

You can fillet parallel lines, xlines, and rays. The current fillet radius temporarily adjusts to create an arc that is tangent to both objects and located in the plane common to both objects.

The first selected object must be a line or a ray, but the second object can be a line, an xline, or a ray. The fillet arc connects as shown in the illustration.

**Fillet Objects with Non-Zero Thickness in 3D**

You can fillet coplanar objects with extrusion directions not parallel to the Z axis of the current UCS. FILLET determines the extrusion direction for the fillet arc in 3D space closest to the direction of the Z axis of the current UCS.

**To set the fillet radius**

1. Click Home tab ➤ Modify panel ➤ Fillet.
2. Enter r (Radius).
3. Enter the fillet radius
4. Select the objects to fillet.
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

FILLET

Rounds and fillets the edges of objects.
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

CHAMFER can be used to bevel all corners of a polyline using a single command.

NOTE Chamfering a hatch boundary that was defined from line segments removes hatch associativity. If the hatch boundary was defined from a polyline, associativity is maintained.

If both objects being chamfered are on the same layer, the chamfer line is created on that layer. Otherwise, the chamfer line is created on the current layer. The layer affects object properties including color and linetype.

Use the Multiple option to chamfer more than one set of objects without leaving the command.

Chamfer by Specifying Distances
The chamfer distance is the amount each object is trimmed or extended to meet the chamfer line or to intersect the other. If both chamfer distances are
0, chamfering trims or extends the two objects until they intersect but does not create a chamfer line. You can hold down Shift while selecting the objects to override the current chamfer distances with a value of 0.

In the following example, you set the chamfer distance to 0.5 for the first line and 0.25 for the second line. After you specify the chamfer distance, you select the two lines as shown.

**Trim and Extend Chamfered Objects**

By default, objects are trimmed when chamfered, but you can use the Trim option to specify that they remain untrimmed.

**Chamfer by Specify Length and Angle**

You can chamfer two objects by specifying where on the first selected object the chamfer line starts, and then the angle the chamfer line forms with this object.

In this example, you chamfer two lines so that the chamfer line starts 1.5 units from the intersection along the first line and forms an angle of 30 degrees with this line.
Chamfer Polylines and Polyline Segments

If the two objects you select for chamfering are segments of a polyline, they must be adjacent or separated by no more than one arc segment. If they are separated by an arc segment, as shown in the illustration, chamfering deletes the arc and replaces it with a chamfer line.

Chamfer an Entire Polyline

When you chamfer an entire polyline, each intersection is chamfered. For best results, keep the first and second chamfer distances equal.

In this example, the chamfer distances are set to equal values.

When you chamfer an entire polyline, only the segments that are long enough to accommodate the chamfer distance are chamfered. The polyline in the following illustration has some segments too short to be chamfered.
To set chamfer distances

1. Click Home tab ➤ Modify panel ➤ Chamfer.
2. Enter d (Distances).
3. Enter the first chamfer distance.
4. Enter the second chamfer distance.
5. Select the lines for chamfering.

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

CHAMFER
Bevels the edges of objects.

CHAMFERA
Sets the first chamfer distance when CHAMMODE is set to 0.

CHAMFERB
Sets the second chamfer distance when CHAMMODE is set to 0.

CHAMFERC
Sets the chamfer length when CHAMMODE is set to 1.

CHAMFERD
Sets the chamfer angle when CHAMMODE is set to 1.
CHAMMODE

Sets the input method for CHAMFER

TRIMMODE

Controls whether selected edges for chamfers and fillets are trimmed.

**Break and Join Objects**

You can break an object into two objects with or without a gap between them. You can also join objects to make a single object.

**Break Objects**

Use BREAK to create a gap in an object, resulting in two objects with a gap between them. BREAK is often used to create space for block or text.

To break an object without creating a gap, specify both break points at the same location. The fastest way to do this is to enter @0,0 at the prompt for the second point.

You can create breaks in most geometric objects except

- Blocks
- Dimensions
- Mlines
- Regions

**Join Objects**

Use JOIN to combine similar objects into a single object. You can also create complete circles and ellipses from arcs and elliptical arcs. You can join

- Arcs
- Elliptical arcs
The object to which you want to join similar objects is called a source object. Objects to be joined must be located in the same plane. Additional restrictions for each type of objects are described in the JOIN command.

**NOTE** When joining two or more arcs (or elliptical arcs), the arcs are joined counterclockwise beginning from the source object.

See also:

- Modify Polylines on page 609
- Modify Complex Objects on page 608

**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 the source object to which you want to join objects.
3. Select one or more objects to join to the source object.
   - Valid objects include arcs, elliptical arcs, lines, polylines, and splines. Additional restrictions for each type of objects are described in the JOIN command.
Quick Reference

BREAK
Breaks the selected object between two points.

JOIN
Joins similar objects to form a single, unbroken object.

Modify Complex Objects
Additional editing operations are available for complex objects, such as blocks, dimensions, hatches, and polylines.

Disassociate Compound Objects (Explode)
You can convert a compound object, such as a polyline, dimension, hatch, or block reference, into individual elements.
You can explode a compound object, such as a polyline, dimension, hatch, or block reference, to convert it into individual elements. For example, exploding a polyline breaks it down to simple lines and arcs. Exploding a block reference or an associative dimension replaces it with copies of the objects that compose the block or dimension.

Explode Dimensions and Hatches
When you explode a dimension or a hatch, all associativity is lost and the dimension or hatch object is replaced by individual objects such as lines, text, points, and 2D solids. To explode dimensions automatically when you create them, set the DIMASSOC system variable to 0.

Explode Polylines
When you explode a polyline, any associated width information is discarded. The resulting lines and arcs follow the polyline’s centerline. If you explode a block that contains a polyline, you need to explode the polyline separately. If you explode a donut, its width becomes 0.

Explode Block References
If you explode a block with attributes, the attribute values are lost, leaving only the attribute definitions. The colors and linetypes of objects in exploded block references can change.
**Explode External References**
An external reference (xref) is a drawing file linked (or attached) to another drawing. You cannot explode xrefs and their dependent blocks.

**To explode an object**

1. Click Home tab ➤ Modify panel ➤ Explode.
2. Select the objects to be exploded.
   For most objects, exploding has no visible effect.

**Quick Reference**

EXPLODE
Breaks a compound object into its component objects.

XPLODE
Breaks a compound object into its component objects.

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.

For example, modify polylines in several ways 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 Multi-Functional Grips**

Polylines are multi-functional, providing context-sensitive options for reshaping the polyline. Polylines have additional multi-functional grips displayed at their midpoints. Control the display of these grips with the GRIPS system variable.

**NOTE** Multi-functional grips also support the standard Grip modes (Stretch, Move, Rotate, Scale, and Mirror). For information on working with multi-functional grips, see *Modify Objects with Multi-Functional Grips* on page 562.

The multi-functional grip-editing options that are available for polylines depend 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 multi-functional grip editing options**

<table>
<thead>
<tr>
<th>Option</th>
<th>Animation</th>
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</thead>
<tbody>
<tr>
<td><strong>Stretch or Stretch Vertex.</strong> Specify a stretch point.</td>
<td></td>
</tr>
<tr>
<td><strong>Add Vertex.</strong> Specify a point for the new vertex.</td>
<td></td>
</tr>
<tr>
<td><strong>Remove Vertex.</strong> Delete the selected vertex.</td>
<td></td>
</tr>
<tr>
<td><strong>Convert to Arc.</strong> Specify the midpoint of a straight segment to convert it to an arc segment.</td>
<td></td>
</tr>
</tbody>
</table>
### Animation Options

<table>
<thead>
<tr>
<th>Option</th>
<th>Animation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Convert to Line</td>
<td>Specify the midpoint of an arc segment to convert into a straight segment.</td>
</tr>
<tr>
<td>Tangent Direction</td>
<td>Manipulate the tangent directions to redefine the shape of a curve-fit polyline.</td>
</tr>
</tbody>
</table>

### Modify Polylines with Coincident Grips

Coincident grips are grips that are shared between multiple objects. When polyline objects sharing the same grip are selected together, multi-functional grip-editing options are not supported for the coincident grip. However, you can still edit the coincident grip using the standard Grip modes (Stretch, Move, Rotate, Scale, and Mirror).

**NOTE** Multi-functional grip-editing options for coincident grips are supported when only one of the polyline objects is selected.

### 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 554
- Use Grip Modes on page 556
- Modify Objects with Multi-Functional Grips on page 562
- Overview of Constraints on page 619
- Trim or Extend Objects on page 586
To modify a polyline

1. Click Home tab ➤ Modify panel ➤ Edit Polyline.
2. Select the polyline to modify.
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.

Enter **x** (Exit) to end a command option. Press Enter to exit the PEDIT command.

**To reverse lines, polylnies, 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

PEDIT
   Edits polylines.
JOIN
   Joins similar objects to form a single, unbroken 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.
GRIPS
   Controls the color of selected grips.
GRIPMULTIFUNCTIONAL

Specifies the access methods to multi-functional grips.

GRIPSUBOBJMODE

Controls whether grips are automatically 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.

Modify Splines

See also:

- **Draw Splines** on page 506
- **Break and Join Objects** on page 606

Modify splines with multi-functional grips or with SPLINEDIT.

Spline editing options are available through both multi-functional grips and SPLINEDIT. These options include add, remove, stretch, refine, and change the tangent direction. SPLINEDIT offers more editing options than the grips, but, to save time, the most commonly-used options are available in the grips.

The grip editing options differ depending on whether the spline is displaying **control vertices (CV's)** or **fit points**. The spline below displays fit point grips on the left, and CV grips on the right.
To switch between the fit point grips and the CV grips, click the triangular grip.

In general, editing a spline with CVs allows you to reshape a small area of the curve, while editing the spline with fit points allows you to reshape the entire curve. Depending on the degree of the curve, the spline only changes through a given set of CVs and you can add more CVs to obtain greater control for a given area of the spline (the \textit{refine} option). Mathematically, fit points affect the entire spline and force the curve to pass through specific points.

To display the grip-editing options, hover over the CV and fit point grips, or select a grip and press Ctrl to cycle through the options. For more information, see \textit{Modify Objects with Multi-Functional Grips} on page 562.

\textbf{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
JOIN
Joins similar objects to form a single, unbroken object.

SPLINE
Creates a smooth curve that passes through fit points or near control vertices.

SPLINEDIT
Edits a spline or spline-fit polyline.

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.
Add Constraints to Geometry

With parametric drawing, you can add constraints to geometry to ensure that the design conforms to specified requirements.

Overview of Constraints

NOTE This topic has been included for AutoCAD-compatibility purposes only. In AutoCAD LT, parametric drawing technology is limited to displaying and hiding constraints, editing constrained geometry, and changing values in the Parameters Manager.

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.

Remove or Relax Constraints

There are two ways to cancel the effects of constraints when you need to make design changes:

- Delete the constraints individually and later apply new constraints. While the cursor hovers over a geometric constraint icon, you can use the Delete key or the shortcut menu to delete the constraint.

- Relax the constraints temporarily on selected objects to make the changes. With a grip selected or when you specify options during an editing command, tap the Ctrl key to alternate between relaxing constraints and maintaining constraints.

Relaxed constraints are not maintained during editing. Constraints are restored automatically if possible when the editing process is complete. Constraints that are no longer valid are removed.

**NOTE** The DELCONSTRAINT command deletes all geometric and dimensional constraints from an object.

Quick Reference

**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.

**LIST**
Displays property data for selected objects.
PARAMETERS
Controls the associative parameters used in the drawing.

-PARAMETERS
Controls the associative parameters used in the drawing.

PARAMETERSCLOSE
Closes the Parameters Manager palette.

TEXTEDIT
Edits a dimensional constraint, dimension, or text object.

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.

Constrain Objects Geometrically
Geometric constraints determine the relationships between 2D geometric objects or points on objects relative to each other.

Overview of Geometric Constraints

NOTE This topic has been included for AutoCAD-compatibility purposes only. In AutoCAD LT, parametric drawing technology is limited to displaying and hiding constraints, editing constrained geometry, and changing values in the Parameters Manager.

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 636
Quick Reference

CONSTRANTBAR
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.

LIST
Displays property data for selected objects.

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.

Apply or Remove Geometric Constraints

Geometric constraints associate geometric objects together, or specify a fixed location or angle.

NOTE
This topic has been included for AutoCAD-compatibility purposes only. In AutoCAD LT, parametric drawing technology is limited to displaying and hiding constraints, editing constrained geometry, and changing values in the Parameters Manager.

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.

<table>
<thead>
<tr>
<th>Constraint</th>
<th>Point</th>
<th>Object</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fix</td>
<td>![Fix Icon]</td>
<td>![Fix Icon]</td>
</tr>
<tr>
<td>Horizontal</td>
<td>![Horizontal Icon]</td>
<td>![Horizontal Icon]</td>
</tr>
<tr>
<td>Vertical</td>
<td>![Vertical Icon]</td>
<td>![Vertical Icon]</td>
</tr>
</tbody>
</table>

The symmetric constraint icons indicate whether it identifying a symmetrical point or object, or the symmetrical line.

<table>
<thead>
<tr>
<th>Constraint</th>
<th>Point</th>
<th>Object</th>
<th>Line</th>
</tr>
</thead>
<tbody>
<tr>
<td>Symmetric</td>
<td>![Symmetric Icon]</td>
<td>![Symmetric Icon]</td>
<td>![Symmetric Icon]</td>
</tr>
</tbody>
</table>

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.

![Diagram of constraint icons]

**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.

Quick Reference

CONSTRANBAR
Displays or hides the geometric constraints on an object.

CONSTRAINTSETTINGS
Controls the display of geometric constraints on constraint bars.

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.

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 constraint 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**

CONSTRRAINTBAR
   Displays or hides the geometric constraints on an object.

CONSTRRAINTSETTINGS
   Controls the display of geometric constraints on constraint bars.
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.

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
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.

For information about temporarily relaxing constraints, see Overview of
Constraints on page 619.

**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 Ctrl key.

5 Move the object. The object moves freely as it is no longer constrained. Constraint bars will no longer be displayed (if enabled) for the object, as the constraints are turned off.

**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**

**CONSTRAINTBAR**
Displays or hides the geometric constraints on an object.

**CONSTRAINTSETTINGS**
Controls the display of geometric constraints on constraint bars.

**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.

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

NOTE This topic has been included for AutoCAD-compatibility purposes only. In AutoCAD LT, parametric drawing technology is limited to displaying and hiding constraints, editing constrained geometry, and changing values in the Parameters Manager.

Dimensional constraints control the size and proportions of a design. They can constrain the following:

- Distances between objects, or between points on objects
- Angles between objects, or between points on objects
- Sizes of arcs and circles

For example, the following illustration includes linear, aligned, angular, and diameter constraints.
If you change the value of a dimensional constraint, all the constraints on the object are evaluated, and the objects that are affected are updated automatically.

Also, constraints can be added directly to segments within a polyline as if they were separate objects.

**NOTE** The number of decimal places displayed in dimensional constraints is controlled by the LUPREC and AUPREC system variables.

**Compare Dimensional Constraints with Dimension Objects**

Dimensional constraints are different from dimension objects in the following ways:

- Dimensional constraints are used in the design phase of a drawing, but dimensions are typically created in the documentation phase

- Dimensional constraints drive the size or angle of objects, but dimensions are driven by objects

- By default, dimensional constraints are not objects, display with only a single dimension style, maintain the same size during zoom operations, and are not plotted

If you need to plot dimensional constraints or use dimension styles, you can change the form of a dimensional constraint from dynamic to annotational. See Apply Dimensional Constraints 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 644.

See also:

- Overview of Geometric Constraints on page 623

Quick Reference

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.

LIST
Displays property data for selected objects.

PARAMETERS
Controls the associative parameters used in the drawing.

-PARAMETERS
Controls the associative parameters used in the drawing.
PARAMETERSCLOSE
Closes the Parameters Manager palette.

TEXTEDIT
Edits a dimensional constraint, dimension, or text object.

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.

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 linewidth.
**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**

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.

LIST
 Displays property data for selected objects.

PARAMETERS
 Controls the associative parameters used in the drawing.

-PARAMETERS
 Controls the associative parameters used in the drawing.

PARAMETERSCLOSE
 Closes the Parameters Manager palette.

TEXTEDIT
 Edits a dimensional constraint, dimension, or text object.

CONSTRAINTNAMEFORMAT
 Controls the text format for dimensional constraints.

CONSTRAINTRELAX
 Indicates whether constraints are enforced or relaxed when editing an object.
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 619.

**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 Ctrl key.
5 Move the object to the desired location.
   The constraint is relaxed for the object, and you should be able to move it.

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

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.

LIST
Displays property data for selected objects.

PARAMETERS
Controls the associative parameters used in the drawing.

-PARAMETERS
Controls the associative parameters used in the drawing.

PARAMETERSCLOSE
Closes the Parameters Manager palette.

TEXTEDIT
Edits a dimensional constraint, dimension, or text object.

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.

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

NOTE This topic has been included for AutoCAD-compatibility purposes only. In AutoCAD LT, parametric drawing technology is limited to displaying and hiding constraints, editing constrained geometry, and changing values in the Parameters Manager.

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**

PARAMETERS

- Controls the associative parameters used in the drawing.

-PARAMETERS

- Controls the associative parameters used in the drawing.

**Control Geometry with the Parameters Manager**

The Parameters Manager displays dimensional constraint parameters (both dynamic constraints and annotational constraints), reference parameters, and user variables.

**NOTE** This topic has been included for AutoCAD-compatibility purposes only. In AutoCAD LT, parametric drawing technology is limited to displaying and hiding constraints, editing constrained geometry, and changing values in the Parameters Manager.

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:

<table>
<thead>
<tr>
<th>Operator</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>+</td>
<td>Addition</td>
</tr>
<tr>
<td>-</td>
<td>Subtraction or unary negation</td>
</tr>
<tr>
<td>%</td>
<td>Floating point modulo</td>
</tr>
<tr>
<td>*</td>
<td>Multiplication</td>
</tr>
<tr>
<td>/</td>
<td>Division</td>
</tr>
<tr>
<td>^</td>
<td>Exponentiation</td>
</tr>
<tr>
<td>(</td>
<td>Parenthesis, expression delimiter</td>
</tr>
<tr>
<td>.</td>
<td>Decimal separator</td>
</tr>
</tbody>
</table>

**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:

<table>
<thead>
<tr>
<th>Function</th>
<th>Syntax</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cosine</td>
<td>( \text{cos}(\text{expression}) )</td>
</tr>
<tr>
<td>Sine</td>
<td>( \text{sin}(\text{expression}) )</td>
</tr>
<tr>
<td>Tangent</td>
<td>( \text{tan}(\text{expression}) )</td>
</tr>
<tr>
<td>Arc cosine</td>
<td>( \text{acos}(\text{expression}) )</td>
</tr>
<tr>
<td>Arc sine</td>
<td>( \text{asin}(\text{expression}) )</td>
</tr>
<tr>
<td>Arc tangent</td>
<td>( \text{atan}(\text{expression}) )</td>
</tr>
<tr>
<td>Hyperbolic cosine</td>
<td>( \text{cosh}(\text{expression}) )</td>
</tr>
<tr>
<td>Hyperbolic sine</td>
<td>( \text{sinh}(\text{expression}) )</td>
</tr>
<tr>
<td>Hyperbolic tangent</td>
<td>( \text{tanh}(\text{expression}) )</td>
</tr>
<tr>
<td>Arc hyperbolic cosine</td>
<td>( \text{acosh}(\text{expression}) )</td>
</tr>
<tr>
<td>Arc hyperbolic sine</td>
<td>( \text{asinh}(\text{expression}) )</td>
</tr>
<tr>
<td>Arc hyperbolic tangent</td>
<td>( \text{atanh}(\text{expression}) )</td>
</tr>
<tr>
<td>Square root</td>
<td>( \text{sqrt}(\text{expression}) )</td>
</tr>
<tr>
<td>Signum function (-1,0,1)</td>
<td>( \text{sign}(\text{expression}) )</td>
</tr>
<tr>
<td>Round to nearest integer</td>
<td>( \text{round}(\text{expression}) )</td>
</tr>
<tr>
<td>Truncate decimal</td>
<td>( \text{trunc}(\text{expression}) )</td>
</tr>
<tr>
<td>Round down</td>
<td>( \text{floor}(\text{expression}) )</td>
</tr>
<tr>
<td>Round up</td>
<td>( \text{ceil}(\text{expression}) )</td>
</tr>
</tbody>
</table>
### Function Syntax

<table>
<thead>
<tr>
<th>Function</th>
<th>Syntax</th>
</tr>
</thead>
<tbody>
<tr>
<td>Absolute value</td>
<td>abs(expression)</td>
</tr>
<tr>
<td>Largest element in array</td>
<td>max(expression1;expression2)</td>
</tr>
<tr>
<td>Smallest element in array</td>
<td>min(expression1;expression2)</td>
</tr>
<tr>
<td>Degrees to radians</td>
<td>d2r(expression)</td>
</tr>
<tr>
<td>Radians to degrees</td>
<td>r2d(expression)</td>
</tr>
<tr>
<td>Logarithm, base $e$</td>
<td>ln(expression)</td>
</tr>
<tr>
<td>Logarithm, base 10</td>
<td>log(expression)</td>
</tr>
<tr>
<td>Exponent, base $e$</td>
<td>exp(expression)</td>
</tr>
<tr>
<td>Exponent, base 10</td>
<td>exp10(expression)</td>
</tr>
<tr>
<td>Power function</td>
<td>pow(expression1;expression2)</td>
</tr>
<tr>
<td>Random decimal, 0-1</td>
<td>Random</td>
</tr>
</tbody>
</table>

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

PARAMETERS
- Controls the associative parameters used in the drawing.
-PARAMETERS
- Controls the associative parameters used in the 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

PARAMETERS
- Controls the associative parameters used in the drawing.

-PARAMETERS
- Controls the associative parameters used in the drawing.

PARAMETERSCLOSE
- Closes the Parameters Manager palette.
Define and Reference Blocks
Work with Blocks

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 blocks. For more information about creating and working with an annotative blocks, see Create Annotative Blocks and Attributes on page 786.

**See also:**

- Scale Annotations on page 764

**Quick Reference**

**BLOCK**

- Creates a block definition from selected objects.

**PURGE**

- Removes unused items, such as block definitions and layers, from the drawing.

**WBLOCK**

- Writes objects or a block to a new drawing file.

**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

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.
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](image)

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 671
- Overview of Blocks on page 655
- Create Block Libraries on page 677
- Work with Dynamic Blocks in Drawings on page 662
- Add Text and Blocks to Tables on page 945
- Add Content with DesignCenter on page 95
- Create and Use Tools from Objects and Images on page 67

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 theGRIPTIPS 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**

**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 and Ctrl-cycling tooltips.

**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.

<table>
<thead>
<tr>
<th>Grip Type</th>
<th>How the Grip Can Be Manipulated in a Drawing</th>
</tr>
</thead>
<tbody>
<tr>
<td>Standard</td>
<td>Within a plane in any direction</td>
</tr>
<tr>
<td>Linear</td>
<td>Back and forth in a defined direction or along an axis</td>
</tr>
<tr>
<td>Rotation</td>
<td>Around an axis</td>
</tr>
<tr>
<td>Flip</td>
<td>Clicked to flip the dynamic block reference</td>
</tr>
<tr>
<td>Alignment</td>
<td>Within a plane in any direction; when moved over an object, triggers the block reference to align with the object</td>
</tr>
<tr>
<td>Lookup</td>
<td>Clicked to display a list of items</td>
</tr>
</tbody>
</table>

**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.

GRIPDYNOCOLOR

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.

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

Controls the associative parameters used in the 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 655

To remove a block definition

1 Click Tools tab ➤ Drawing Utilities panel ➤ Purge. The Purge dialog box displays a tree view of named objects that can be purged.

2 To purge blocks, use one of the following methods:
   ■ To purge all unreferenced blocks, select Blocks. To include nested blocks, select Purge Nested Items.
   ■ To purge specific blocks, double-click Blocks to expand the Block tree view. Select the blocks to be purged.
   If the item you want to purge is not listed, select View Items You Cannot Purge.

3 You are prompted to confirm each item in the list. If you do not want to confirm each purge, clear the Confirm Each Item to Be Purged option.

4 Click Purge.
To confirm the purging of each item, respond to the prompt by choosing Yes or No, or Yes to All if more than one item is selected.

Select more items to purge, or click Close.

Quick Reference

PURGE

Removes unused items, such as block definitions and layers, from the drawing.
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 655.
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 the Block Editor* on page 706

**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 1173.

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.
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.

<table>
<thead>
<tr>
<th>If you want objects in a block to</th>
<th>Create objects on these layers</th>
<th>Create objects with these properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>Retain original properties</td>
<td>Any but 0 (zero)</td>
<td>Any but BYBLOCK or BYLAYER</td>
</tr>
<tr>
<td>Inherit properties from the current layer</td>
<td>0 (zero)</td>
<td>BYLAYER</td>
</tr>
<tr>
<td>Inherit individual properties first, then layer properties</td>
<td>Any</td>
<td>BYBLOCK</td>
</tr>
</tbody>
</table>

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 305

**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.

See also:
- Add Constraints to Dynamic Blocks

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 that can be viewed in DesignCenter.

Optionally, you can also document each block definition by inserting it in the drawing area of the library drawing. In addition to the block geometry, you can include text that provides the block name, the date of creation, the date of the last modification, and any special instructions or conventions. This creates a visual index of the blocks in the block library drawing.

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 (ADCENT).
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 67.

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 508
- Add Constraints to Geometry on page 619

**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 attributes. For more information about creating and working with an annotative attributes, see Create Annotative Blocks and Attributes on page 786.

See also:

- Modify the Data in Block Attributes on page 694
- Scale Annotations on page 764
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 display of the in-place editor used to create multiline attributes.

ATTMODE
Controls display of attributes.

ATTMULTI
Controls whether multiline attributes can be created.

ATTREQ
Controls whether INSERT uses default attribute settings during insertion of blocks.
Define Block Attributes

The characteristics include the tag, which is a name that identifies the attribute, the prompt displayed when you insert the block, value information, text formatting, location within the block, and any optional modes (Invisible, Constant, Verify, Preset, Lock Position, and Multiple Lines).

If you plan to extract the attribute information for use in a parts list, you may want to keep a list of the attribute tags you have created. You will need this tag information later when you create the attribute template file.

Choose Attribute Modes

Attribute modes control the behavior of attributes in blocks. For example, you can control

■ Whether an attribute is visible or invisible in the drawing
■ Whether an attribute has a constant value, such as a part number
■ Whether the attribute can be moved relative to the rest of the block
■ Whether the attribute is a single-line attribute or a multiple-line attribute

If an attribute has a constant value, you will not be prompted for its value when you insert the block. If an attribute has a variable value, such as the asset number of a computer, you will be prompted when you insert the block.

Understand Single-Line and Multiple-Line Attributes

There are several differences between single-line and multiple-line attributes.

■ Single-line attributes are limited to 255 characters from the user interface.
■ Multiple-line attributes provide more formatting options than single-line attributes.
■ When editing single-line and multiple line attributes, different editors are displayed.
■ Multiple line attributes display four grips similar to MTEXT objects, while single-line attributes display only one grip.
■ When a drawing is saved to AutoCAD LT 2007 or earlier, a multiple-line attribute is converted to several single-line attributes, one for every line of text in the original multiple-line attribute. If the drawing file is opened in
the current release, these single line attributes are automatically merged back into a multiple-line attribute.

**NOTE** If a multiple-line attribute makes a round trip to an earlier release of AutoCAD LT, 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 LT displays a dialog box that lets you cancel the operation.

**Correct Mistakes in Block Attribute Definitions**

If you make a mistake, you can use the Properties palette or DDEDIT to make limited changes to an attribute definition before it is associated with a block. If you need to make more extensive changes, delete the attribute definition and create a new one.

**Attach Attributes to Blocks**

After you create one or more attribute definitions, you attach the attributes to a block when you define or redefine that block. When you are prompted to select the objects to include in the block definition, include in the selection set any attributes you want to attach to the block.

To use several attributes together, define them and then include them in the same block. For example, you can define attributes tagged "Type," "Manufacturer," “Model,” and “Cost,” and then include them in a block called CHAIR.

![Block Example](image.png)

Usually, the order of the attribute prompts is the same as the order in which you selected the attributes when you created the block. However, if you used crossing or window selection to select the attributes, the order of the prompts is the reverse of the order in which you created attributes. You can use the Block Attribute Manager to change the order in which you are prompted for attribute information when you insert the block reference.

When you open a block definition in the Block Editor, you can use the Attribute Order dialog box to change the order in which you are prompted for attribute information when you insert the block reference.
Use Attributes Without Attaching Them to Blocks

Stand-alone attributes can also be created. Once attributes have been defined, and the drawing is saved, this drawing file can be inserted into another drawing. When the drawing is inserted, you are prompted for the attribute values.

To create an attribute definition

1. Click Home tab ➤ Block panel ➤ Define Attributes.
2. In the Attribute Definition dialog box, set the attribute modes and enter tag information, location, and text options.
3. (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 display of the in-place editor used to create multiline attributes.
ATTMULTI
   Controls whether multiline attributes can be created.
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 LT, 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 Cwww000 (Block name)
BL:LEVEL Nwww000 (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 Nwww000 (Block counter; the same for MINSERT)
BL: HANDLE Cwww000 (Block handle; the same for MINSERT)
BL: LAYER Cwww000 (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)
characterCwww000 (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.

<table>
<thead>
<tr>
<th>Field</th>
<th>(C)haracter or (N)umeric data</th>
<th>Maximum field length</th>
<th>Decimal places</th>
</tr>
</thead>
<tbody>
<tr>
<td>Block name</td>
<td>C</td>
<td>040</td>
<td>000</td>
</tr>
<tr>
<td>Manufacturer</td>
<td>C</td>
<td>006</td>
<td>000</td>
</tr>
<tr>
<td>Model</td>
<td>C</td>
<td>015</td>
<td>000</td>
</tr>
<tr>
<td>Cost</td>
<td>N</td>
<td>006</td>
<td>002</td>
</tr>
</tbody>
</table>
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.d)
```

**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.

<table>
<thead>
<tr>
<th>(NAME)</th>
<th>(X)</th>
<th>(Y)</th>
<th>(SUPPLIER)</th>
<th>(MODEL)</th>
<th>(PRICE)</th>
</tr>
</thead>
<tbody>
<tr>
<td>DESK</td>
<td>120.0</td>
<td>49.5</td>
<td>ACME IN-DUST.</td>
<td>51-793W</td>
<td>379.95</td>
</tr>
<tr>
<td>CHAIR</td>
<td>122.0</td>
<td>47.0</td>
<td>ACME IN-DUST.</td>
<td>34-902A</td>
<td>199.95</td>
</tr>
<tr>
<td>DESK</td>
<td>-77.2</td>
<td>40.0</td>
<td>TOP DRAWER INC.</td>
<td>X-52-44</td>
<td>249.95</td>
</tr>
</tbody>
</table>

The order of the fields corresponds to the order of the fields in the template files. You can use these files in other applications, such as spreadsheets, and you can sort and manipulate the data as needed. For example, you can open an attribute extraction file in Microsoft Excel in which you can specify a separate column for each field. See the documentation for your spreadsheet program for information about how to use data from other applications. If you open the file in Notepad or another Windows text processor, you can paste the information back into the drawing as text.

**Nested Blocks**

The line BL:LEVEL in a template file reports the nesting level of a block reference. A block that is inserted in a drawing has a nesting level of 1. A block reference that is part of (nested within) another block has a nesting level of 2, and so on.

For a nested block reference, the $X$, $Y$, $Z$ coordinate values, scale factors, extrusion direction, and rotation angle reflect the actual location, size, orientation, and rotation of the nested block in the world coordinate system.

In some complex cases, nested block references cannot be correctly represented with only two scale factors and a rotation angle, for example, if a nested block reference is rotated in 3D. When this happens, the scale factors and rotation angle in the extracted file record are set to zero.

**Error Handling**

If a field is not wide enough for the data that is to be placed in it, the data is truncated and the following message is displayed:

** Field overflow in record <record number>
This could happen, for example, if you have a BL:NAME field with a width of 8 characters and a block in your drawing has a name 10 characters long.

To create an attribute extraction template file

1. Start Notepad.
   You can use any text editor or word processor that can save a text file in ASCII format.

2. Enter template information in Notepad. See Extract Block Attribute Data (Advanced) on page 686 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.
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.
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 679

■ Modify Dynamic Block Definitions on page 712

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 Edit Attributes dialog box
■ 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.

See also:
■ Modify a Block Definition on page 692

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 display of the in-place editor used to create multiline attributes.

ATTMULTI
Controls whether multiline attributes can be created.
Modify a Block Attribute Definition

You can modify attributes in block definitions with the Block Attribute Manager. For example, you can modify the following:

- Properties that define how values are assigned to an attribute and whether or not the assigned value is visible in the drawing area
- Properties that define how attribute text is displayed in the drawing
- Properties that define the layer that the attribute is on and the attribute line's color, weight, and type

By default, attribute changes you make are applied to all existing block references in the current drawing.

Changing the attribute properties of existing block references does not affect the values assigned to those blocks. For example, in a block containing an attribute whose tag is Cost and value is 19.99, the 19.99 value is unaffected if you change the tag from Cost to Unit Cost.

Updating attributes with duplicate tag names can lead to unpredictable results. Use the Block Attribute Manager to find duplicate tags and change tag names.

If constant attributes or nested attributed blocks are affected by your changes, use REGEN to update the display of those blocks in the drawing area.

Change the Prompt Order for Attribute Values

When you define a block, the order in which you select the attributes determines the order in which you are prompted for attribute information when you insert the block. You can use the Block Attribute Manager to change the order of prompts that request attribute values.

Remove Block Attributes

You can remove attributes from block definitions and from all existing block references in the current drawing. Attributes removed from existing block references do not disappear in the drawing area until you regenerate the drawing using REGEN.

You cannot remove all attributes from a block; at least one attribute must remain. If you need to remove all attributes, redefine the block.
Update Block References

You can update attributes in all block references in the current drawing with changes you made to the block definition. For example, you may have used the Block Attribute Manager to modify attribute properties in several block definitions in your drawing but elected not to automatically update existing block references when you made the changes. Now that you are satisfied with the attribute changes you made, you can apply those changes to all blocks in the current drawing.

You can also use ATTSYNC to update attribute properties in block references to match their block definition, or to update a block instance after you redefine a block attribute using BLOCK, -BLOCK, or BEDIT.

Updating attribute properties in block references does not affect any values that have been assigned to those attributes.

Edit Attributes in a Block Reference

You can select an attribute in a block reference and use the Properties palette to change its properties, or you can use the Enhanced Attribute Editor to modify all the attributes in a selected block reference.

See also:

- Define Block Attributes on page 682
- Modify a Block Definition on page 692

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.
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 display of the in-place editor used to create multiline attributes.
ATTMULTI

Controls whether multiline attributes can be created.

Disassemble a Block Reference (Explode)

If you need to modify one or more objects within a block separately, you can disassemble, or explode, the block reference into its component objects. After making the changes, you can

- Create a new block definition
- Redefine the existing block definition
- Leave the component objects uncombined for other uses

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)

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 dimensions

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 of 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

Plan the block content.

Know how the block should change or move, and what parts will depend on the others.
Example: The block will be resizable, and after it is resized, additional geometry is displayed.

Draw the geometry.

Draw the block geometry in the drawing area or the Block Editor.
NOTE If you will use visibility states to change how geometry is displayed, you may not want to include all the geometry at this point. For more information, see Control the Visibility of Objects in a Block on page 747.

Add parameters.

Add either individual parameters or parameter sets to define geometry that will be affected by an action or manipulation. Keep in mind the objects that will be dependent on one another.
Add actions.

If you are working with action parameters on page 719, if necessary, add actions to define what will happen to the geometry when it is manipulated.

Define custom properties.

Add properties that determine how the block is displayed in the drawing area. Custom properties affect grips, labels, and preset values for block geometry.

Test the block.

On the ribbon, in the Block Editor contextual tab, Open/Save panel, click Test Block to test the block before you save it.

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 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 682

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 \texttt{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 718
- Overview of Actions on page 719
- Control Tool Properties on page 76
- Organize Tool Palettes on page 84
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 LT 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

![Alert Icon]

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 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 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.

<table>
<thead>
<tr>
<th>Selected object in the Block Editor</th>
<th>Objects that are dependency highlighted</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parameter</td>
<td>Associated grips and actions</td>
</tr>
<tr>
<td>Grip</td>
<td>Associated parameter and actions</td>
</tr>
<tr>
<td>Action</td>
<td>Associated parameters, grips, and the selection set (geometry)</td>
</tr>
</tbody>
</table>

**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.
At the Command prompt, enter `bactioncolor`.

Enter one of the following values:

- `BYLAYER`
- `BYBLOCK`
- An integer from 1 to 255.

Press Enter.

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 parameters 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.
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.

LOOKUPTABLE

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.

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 728.

**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 Block Attribute Data (Advanced) on page 686

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.

<table>
<thead>
<tr>
<th>Property name</th>
<th>Description</th>
<th>Associated parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>Angle Name</td>
<td>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.</td>
<td>Point, Linear, Polar, XY, Rotation, Flip, Visibility, Lookup</td>
</tr>
<tr>
<td>Distance Name</td>
<td></td>
<td>Linear, Flip</td>
</tr>
<tr>
<td>Flip Name</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Hor Distance Name</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Lookup Name</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Position Name</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ver Distance Name</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Visibility Name</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Property name</td>
<td>Description</td>
<td>Associated parameter</td>
</tr>
<tr>
<td>-----------------------</td>
<td>-----------------------------------------------------------------------------</td>
<td>----------------------</td>
</tr>
<tr>
<td>Angle Description</td>
<td>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.</td>
<td>Point, Linear, Polar</td>
</tr>
<tr>
<td>Distance Description</td>
<td></td>
<td>XY, Rotation, Flip</td>
</tr>
<tr>
<td>Flip Description</td>
<td></td>
<td>Visibility, Lookup</td>
</tr>
<tr>
<td>Hor Distance Description</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Position Description</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ver Distance Description</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Visibility Description</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Lookup Description</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Base State Name</td>
<td>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.</td>
<td>Flip</td>
</tr>
<tr>
<td>Flipped State Name</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Base Type</td>
<td>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.</td>
<td>Rotate</td>
</tr>
<tr>
<td>Distance Type</td>
<td>Specifies whether the distance applied to the move is the parameter's X value, Y value, or X and Y coordinate value from the parameter's base point.</td>
<td>Move, Stretch</td>
</tr>
<tr>
<td>Show Properties</td>
<td>Specifies whether custom properties will be displayed in the Properties palette for the selected block reference.</td>
<td>Point, Linear, Polar</td>
</tr>
<tr>
<td>Property name</td>
<td>Description</td>
<td>Associated parameter</td>
</tr>
<tr>
<td>---------------</td>
<td>-------------</td>
<td>----------------------</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ XY</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ Rotate</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ Flip</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ Visibility</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ Lookup</td>
</tr>
</tbody>
</table>

See also:

■ Define User Parameters in Dynamic Blocks
■ Specify Value Sets for Dynamic Blocks on page 738
■ Use Chained Actions on page 731
■ Specify Grips for Dynamic Blocks on page 741

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 720
- Stretch on page 723
- Polar Stretch on page 724

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

<table>
<thead>
<tr>
<th>Parameter Type</th>
<th>Grip Type</th>
<th>Actions You Can Associate with a Parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>Point</td>
<td>Standard</td>
<td>Move, Stretch</td>
</tr>
<tr>
<td>Linear</td>
<td>Linear</td>
<td>Move, Scale, Stretch, Array</td>
</tr>
<tr>
<td>Polar</td>
<td>Standard</td>
<td>Move, Scale, Stretch, Polar Stretch, Array</td>
</tr>
</tbody>
</table>

Specify Distance and Angle Values | 741
<table>
<thead>
<tr>
<th>Parameter Type</th>
<th>Grip Type</th>
<th>Actions You Can Associate with a Parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>XY</td>
<td>Standard</td>
<td>Move, Scale, Stretch, Array</td>
</tr>
<tr>
<td>Rotation</td>
<td>Rotation</td>
<td>Rotate</td>
</tr>
<tr>
<td>Flip</td>
<td>Flip</td>
<td>Flip</td>
</tr>
<tr>
<td>Alignment</td>
<td>Alignment</td>
<td>None (The action is implied and contained within the parameter.)</td>
</tr>
<tr>
<td>Visibility</td>
<td>Lookup</td>
<td>None (The action is implied and controlled by visibility states.)</td>
</tr>
<tr>
<td>Lookup</td>
<td>Lookup</td>
<td>Lookup</td>
</tr>
<tr>
<td>Base</td>
<td>Standard</td>
<td>None</td>
</tr>
</tbody>
</table>

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 747 or a *lookup table* on page 727, 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.

<table>
<thead>
<tr>
<th>Grip Type</th>
<th>How the Grip Can Be Manipulated in a Drawing</th>
<th>Associated Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>Standard</td>
<td>Within a plane in any direction</td>
<td>Base, Point, Polar, and XY</td>
</tr>
<tr>
<td>Linear</td>
<td>Back and forth in a defined direction or along an axis</td>
<td>Linear</td>
</tr>
<tr>
<td>Rotation</td>
<td>Around an axis</td>
<td>Rotation</td>
</tr>
<tr>
<td>Flip</td>
<td>Click to flip the dynamic block reference</td>
<td>Flip</td>
</tr>
<tr>
<td>Alignment</td>
<td>Within a plane in any direction; when moved over an object, triggers the block reference to align with the object</td>
<td>Alignment</td>
</tr>
<tr>
<td>Lookup</td>
<td>Click to display a list of items</td>
<td>Visibility, Lookup</td>
</tr>
</tbody>
</table>
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.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Point</td>
<td>Position description</td>
</tr>
<tr>
<td>Linear</td>
<td>Distance description</td>
</tr>
<tr>
<td>Polar</td>
<td>Distance description, angle description</td>
</tr>
<tr>
<td>XY</td>
<td>Horizontal distance description, vertical distance description</td>
</tr>
<tr>
<td>Rotation</td>
<td>Angle description</td>
</tr>
<tr>
<td>Flip</td>
<td>Flip description</td>
</tr>
<tr>
<td>Visibility</td>
<td>Visibility description</td>
</tr>
<tr>
<td>Lookup</td>
<td>Lookup description</td>
</tr>
<tr>
<td>Alignment</td>
<td>Set to “Aligns block to object”</td>
</tr>
<tr>
<td>Basepoint</td>
<td>No special tooltip is required</td>
</tr>
</tbody>
</table>

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.
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 and Ctrl-cycling tooltips.

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.

Control the Visibility of Objects in a Block | 749
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.
Work with 3D Models
Create 3D Models

You can display 3D objects from products such as AutoCAD in AutoCAD LT. Creation of 3D objects is limited to objects with 3D thickness or objects that can be displayed using the wireframe visual style.

Create Wireframe Models

A wireframe model is an edge or skeletal representation of a real-world 3D object using lines and curves.

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 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:

- Enter 3D coordinates that define the X, Y, and Z location of the object.
- Set the default workplane (the XY plane of the UCS) on which to draw the object.
- Move or copy the object to its proper 3D location after you create it.

Wireframe modeling is a skill that requires practice and experience. The best way to learn how to create wireframe models is to begin with simple models before attempting models that are more complex.

Tips for Working with Wireframe Models

Creating 3D wireframe models can be more difficult and time-consuming than creating their 2D views. Here are some tips that will help you work more effectively:

- Plan and organize your model so that you can turn off layers to reduce the visual complexity of the model. Color can help you differentiate between objects in various views.
- Create construction geometry to define the basic envelope of the model.
- Use multiple views, especially isometric views, to make visualizing the model and selecting objects easier.
- Become adept at manipulating the UCS in 3D. The XY plane of the current UCS operates as a workplane to orient planar objects such as circles and arcs. The UCS also determines the plane of operation for trimming and extending, offsetting, and rotating objects.
- Use object snaps and grid snap carefully to ensure the precision of your model.
- Use coordinate filters to drop perpendiculars and easily locate points in 3D based on the location of points on other objects.

To drop a perpendicular line from a 3D point down to the XY plane

1. Click Home tab ▶ Draw panel ▶ Line.
2 Use an object snap to specify a point on an object that is not on the XY plane of the UCS.
   This location defines the first point of the line.

3 Enter .xy and then enter @ at the Of prompt.
   This operation extracts the X and Y coordinate values from the first point.

4 Enter 0 to specify the Z value.
   The X and Y values extracted from the first point are combined with a new Z value (0) to finish the definition of the second point.

5 Press Enter to end the command.

**Quick Reference**

**Commands**

**UCS**

Manages user coordinate systems.

**System Variables**

**ELEVATION**

Stores the current elevation of new objects relative to the current UCS.

**FACETRES**

**ISOLINES**

Specifies the number of contour lines per surface on objects.

**Add 3D Thickness to Objects**

Use the thickness property to give objects a 3D appearance.

The 3D thickness of an object is the distance that object is extended, or thickened, above or below its location in space. Positive thickness extrudes upward in the positive Z direction; negative thickness extrudes downward (negative Z). Zero (0) thickness means that there is no 3D thickening of the object.
The orientation of the UCS when the object was created determines the $Z$ direction. Objects with a non-zero thickness can be shaded and can hide other objects behind them.

The thickness property changes the appearance of the following types of objects:

- 2D solids
- Arcs
- Circles
- Lines
- Polylines (including spline-fit polylines, rectangles, polygons, boundaries, and donuts)
- Text (only if created as a single-line text object using an SHX font)
- Traces
- Points

Modifying the thickness property of other types of objects does not affect their appearance.

You can set the default thickness property for new objects you create by setting the THICKNESS system variable. For existing objects, change the thickness property on the Properties palette.

The 3D thickness is applied uniformly to an object; a single object cannot have different thicknesses.

You might need to change the 3D viewpoint to see the effect of thickness on an object.

**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.

System Variables
THICKNESS
Sets the current 3D thickness.
VIEWMODE
Stores the View mode for the current viewport.
Annotate Drawings
Work with Annotations

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.

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*. This property allows you to automate the process of scaling annotations so that they plot or display at the correct size on the paper.

Instead of creating multiple annotations at different sizes and on separate layers, you can turn on the annotative property by object or by style, and set the annotation scale for layout or model viewports. The annotation scale controls the size of the annotative objects relative to the model geometry in the drawing.

The following objects are commonly used to annotate drawings and contain an annotative property:

- Text
- Dimensions
- Hatches
- Tolerances
- Multileaders
- Blocks
- Attributes

When the Annotative property for these objects is turned on (set to Yes), these objects are called *annotative objects*.

You define a paper size for annotative objects. The *annotation scale* you set for layout viewports and model space determines the size of the annotative objects in those spaces.

Save to Legacy Drawing File Format

Set the system variable SAVEFIDELITY to 1 when you save a drawing that contains annotative objects to a legacy drawing file format (AutoCAD LT 2007 or earlier). This preserves the visual fidelity of the drawing when it is opened in a release earlier than AutoCAD LT 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 LT 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 179.
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 774.
2. In model space, set the annotation scale to the scale at which the annotations will be plotted or displayed on page 767.
3. Create annotative objects using annotative styles on page 774.

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 793.
2. Set the annotation scale to the new scale on page 768 (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 263 or make a layout current on page 271.
2. Create viewports on page 280.
3. Set the annotation scale for each viewport on page 768. (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 791. For more information about adding scales to annotative objects, see Add and Modify Scale Representations on page 793.

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

is a setting that is saved with model space, layout viewports, and model views. When you add objects to your drawing, they support the current annotation scale and are scaled based on that scale setting and automatically displayed at the correct size in model space.

Before you add annotative objects to your model, you set the annotation scale. Think about the eventual scale settings of the viewports in which the annotations will display. The annotation scale should be set to the same scale as the viewport in which the annotative objects will display in the layout (or the plot scale if plotting from model space). For example, if the annotative objects will display in a viewport that has a scale of 1:2, then you set the annotation scale to 1:2.

When working on the model tab or when a viewport is selected, the current annotation scale is displayed on the application or drawing status bar. You can use the status bars to change the annotation scale. You can reset the annotation scale list to the default list of scales defined in the registry 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 drawing status bar or application 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 the registry are merged into the drawing.

See also:
■ Drawing Status Bar on page 50

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*. When the Annotative property for these objects is turned on (set to Yes), these objects are called *annotative objects*.

Overview of Creating Annotative Objects
When you add annotations to your drawing, you can turn on the property for those objects. These annotative objects are scaled based on the current setting and are automatically displayed at the correct size.

Annotative objects are defined at a paper height and display at the size determined by the annotation scale.

The following objects can be annotative (have an Annotative property):

- Hatches
- Text (single-line and multiline)
- Dimensions
- Tolerances
- Leaders and multileaders (created with MLEADER)
- Blocks
- Attributes
Many of the dialog boxes used to create these objects contain an Annotative check box where you can make the object annotative. You can also change existing objects to be annotative by changing the annotative property in the Properties palette.

When you hover the cursor over an annotative object that supports one annotation scale, the cursor displays a icon. When the object supports more than one annotation scale, it displays a icon.
Text, dimension, and multileader styles can also be annotative. Annotative styles create annotative objects.

**Visual Fidelity for Annotative Objects**

When working with objects, this option allows you to maintain visual fidelity for these objects when they are viewed in AutoCAD LT 2007 and earlier releases. Visual fidelity is controlled by the SAVEFIDELITY system variable.

If you work primarily in model space, it is recommended that you turn off visual fidelity (set SAVEFIDELITY to 0). However, if you need to exchange drawings with other users, and layout fidelity is most important, then visual fidelity should be turned on (set SAVEFIDELITY to 1).

**NOTE** The SAVEFIDELITY system variable does not affect saving a drawing to the AutoCAD LT 2010 drawing or DXF file formats.

Annotative objects may have multiple . When visual fidelity is on, annotative objects are decomposed and scale representations are saved (in an ) to separate layers, which are named based on their original layer and appended with a number. If you explode the block in AutoCAD LT 2007 or earlier releases, and then open the drawing in AutoCAD LT 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 LT 2008 and later releases in AutoCAD LT 2007 and earlier releases.
When this option is not selected, a single model space representation is displayed on the Model tab. More annotation objects may be displayed on the Model tab depending on the ANNOALLVISIBLE setting. Also, more objects may be displayed in paper space viewports at different sizes than in AutoCAD LT 2008 and later releases.

For a procedure to set this option for annotative objects, see To save drawings with visual fidelity for annotative objects.

See also:
- Work with Annotative Styles on page 772

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.

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 objects.
The dialog boxes used to define these objects contain an Annotative check box where you can make the styles annotative. Annotative styles display a special icon before their names in dialog boxes and the Properties palette.

You should specify the Paper Height value for any annotative text styles you create. The Paper Height setting specifies the height of the text in paper space.

**NOTE** If you’ve specified the Paper Height value for a dimension or multileader style, this setting overrides the text style Paper Height setting.

If you redefine styles to be annotative or nonannotative, existing objects that reference those styles are not automatically updated to reflect the annotative property of the style or definition. Use the ANNOUPDATE command to update the existing objects to the current Annotative properties of the style.

When you change the Style property of an existing object (whether it’s annotative or nonannotative), the object’s annotative properties will match that of the new style. If the style does not have a fixed height (the Height value is 0), the paper height of the object is calculated based on the object’s current height and the annotation scale.
See also:
- Work with Text Styles on page 901
- Create Annotative Text on page 775
- Use Dimension Styles on page 961
- Create Annotative Dimensions and Tolerances on page 779
- Work with Leader Styles on page 886
- Create Annotative Leaders and Multileaders on page 784

To create an annotative style
- Follow the steps in one of the following procedures
  - To create a new annotative text style
  - To change an existing nonannotative text style to annotative
  - To create a new annotative dimension style
  - To change an existing dimension style to annotative
  - To create a new annotative multileader style
  - To change an existing multileader style to annotative

To create annotative objects from annotative styles
- Follow the steps in one of the following procedures
  - To create annotative single-line text
  - To create annotative multiline text
  - To create an annotative dimension
  - To create an annotative multileader
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 text for notes and labels in your drawing. You create annotative text by using an annotative text style, which sets the height of the text on the paper. The current automatically determines the display size of the text in model space or paper space viewports.

For example, you want text to display at a height of 3/16" on the paper, so you can define a text style to have a paper height of 3/16". When you add text to a viewport that has a scale of 1/2"=1'0", the current annotation scale, which is set to the same scale as the viewport’s, automatically scales the text to display appropriately at 4.5".

You can also change existing nonannotative text to annotative by changing the text's Annotative property to Yes. This applies to any text created through text styles or through the TEXT and MTEXT commands.

You can set the orientation of annotative text objects to match the orientation of the paper. For more information about setting the orientation of annotative objects, see Set Orientation for Annotations on page 796.

See also:

- Create Text on page 840
- Work with Annotative Styles on page 772
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 nonannotative text style to annotative

1. Click Annotate tab ➤ Text panel ➤ Text Style.
2. In the Text Style dialog box, Styles list, select a style.

NOTE A icon next to a text style name indicates that the style is already annotative.

3. Under Size, select Annotative.
4. In the Paper Text Height box, enter the height of the text as it will display on paper.
5. Click Apply.
6. (Optional) Click Set Current to set this style as the current text style.
7. Click Close.
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 nonannotative
1 Double-click a multiline text object. The In-Place Text Editor is displayed.

2 Click the Annotative \button on the toolbar to change existing multiline text to annotative or nonannotative. When the Annotative button is depressed, the text is annotative. When the button is not depressed, the text is nonannotative.

3 Click OK to save the changes.

To change existing text (single-line or multiline) to be annotative or nonannotative
1 In the drawing, select a text object.

2 Click View tab ➤ Palettes panel ➤ Properties.

3 In the Properties palette, under Text, click Annotative.

4 On the drop-down list, select Yes or No.

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.

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 dimensions for measurements in your drawing through annotative dimension styles.

Annotative dimension styles create dimensions in which all the elements of the dimension, such as text, spacing, and arrows, scale uniformly by the .
If you associate a dimension to an annotative object, the associativity of the dimension is lost.

You can also change an existing nonannotative dimension to annotative by changing the dimension’s Annotative property to Yes.

**NOTE** When the current dimension style is annotative, the value of DIMSCALE is automatically set to zero, and does not affect the dimension scale.

You can also create annotative tolerances. Geometric tolerances show acceptable deviations of form, profile, orientation, location, and runout of a feature.

**See also:**
- Dimensions and Tolerances on page 955
- Use Dimension Styles on page 961
- Work with Annotative Styles on page 772

**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.
In the New Dimension Style dialog box, select the appropriate tab and make changes to define the dimension style.

Click OK.

(Optional) Click Set Current to set this style as the current dimension style.

Click Close.

To change an existing dimension style to annotative

Click Annotate tab ➤ Dimensions panel ➤ Dimension Style.

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.

Click Modify.

In the Modify Dimension Style dialog box, Fit tab, under Scale for Dimension Features, select Annotative.

Click OK.

(Optional) Click Set Current to set this style as the current dimension style.

Click Close.

To create an annotative dimension

Click Annotate tab ➤ Dimensions panel ➤ Dimension Style.

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

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one first. For information on creating an annotative dimension style see, To create a new annotative dimension style.

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 nonannotative

1 Select a dimension in a drawing.

2 Click View tab ➤ Palettes panel ➤ Properties.
3 In the Properties palette, under Misc, click Annotative.
4 On the drop-down list, select Yes or No.

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.

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 multileader on page 1401 are used to add call outs to your drawings. You can create leaders through an annotative dimension style and multileaders through an annotative multileader style.

When you create a leader, you create two separate objects: the leader and the text, block, or tolerance associated with the leader. When you create a multileader, you create a single object.

If the multileader style is annotative, the associated text or tolerance will be annotative as well, regardless of the annotative setting of the text style or tolerance.

NOTE It is recommended that you create non-annotative entities when creating a mleader content block.

Blocks used in leaders and multileaders must be non-annotative.

You can change the Annotative property of leaders and multileaders in the Properties palette.

See also:

- Create Leaders on page 876
- Work with Leader Styles on page 886

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 nonannotative
1  Select a leader or multileader in a drawing.

2  Click View tab ➤ Palettes panel ➤ Properties.
3  In the Properties palette, under Misc, click Annotative.
4  On the drop-down list, select Yes or No.

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.
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.

![Image]

You cannot change the Annotative property of individual block references. To set an annotative block’s paper size, you should define the block in paper space or on the Model tab with the set to 1:1.

When creating and working with annotative blocks and annotative objects within blocks, the following points should be noted:

- Nonannotative blocks can contain annotative objects, which are scaled by the block’s scale factor in addition to the annotation scale.

- Annotative blocks cannot reside in annotative blocks.

- Annotative block references are scaled uniformly by the current annotation scale as well as any user scale applied to the block reference.

- Blocks that contain annotative objects should not be manually scaled.

You can define annotative attributes for annotative and nonannotative blocks. Use annotative attributes with nonannotative blocks when you want the geometry in the block to display on the paper based on the scale of the viewport, but you want the attribute text to display at the Paper Height defined for the attribute.
You can set the orientation of annotative blocks to match the orientation of the paper. For more information about setting the orientation of annotative objects, see Set Orientation for Annotations on page 796.

You can use the ANNOTATIVEDWG system variable to specify whether or not the entire drawing will behave as an annotative block when inserted into another drawing. The ANNOTATIVEDWG system variable becomes read-only if the drawing contains annotative objects.

**NOTE** The INSUNITS setting is ignored when inserting blocks into a drawing.

**See also:**
- Work with Blocks on page 655
- Attach Data to Blocks (Block Attributes) on page 679

**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 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 aclt.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 dialog box should be selected.

You can use the HPANNOTATIVE system variable to specify whether or not new hatches are annotative. By default, new hatch objects are nonannotative.

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 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

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 50

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 object in your drawing, it supports one scale: 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. 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

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 901
- Create Annotative Text on page 775
- Create Annotative Blocks and Attributes on page 786
- Create Annotative Hatches on page 789

To match the layout's orientation for an annotative text style

1. Click Annotate tab ➤ Text panel ➤ Text Style.
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.

Under Size, select Match Text Orientation to Layout.

Click Apply.

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

- Overview of Hatch Patterns and Fills on page 799
- Specify Hatch and Fill Areas on page 807
- Control the Appearance of Hatches on page 812
- Modify Hatches and Fills on page 829
- Create a Blank Area to Cover Objects on page 835

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.

![Diagram of hatch pattern](image)

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.

![Diagram of user coordinate system](image)

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.

Alternatively, solid-filled areas can be created using

- 2D solids (SOLID)
- Wide polylines (PLINE)
- Donuts (DONUT)

**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
Manages user coordinate systems.

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.

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 to display a preview when specifying internal points for a hatch.

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 -HATCH Draw option.
- 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 the HATCH command. They are removed when you specify another internal point for the hatch, or when you use the REDRAW, REGEN, or REGENALL commands.
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 with the HPQUICKPREVIEW system variable.

**See also:**
- Reshape a Hatch or Fill on page 831

**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.
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 to display a preview when specifying internal points for a hatch.

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 `acadlt.pat` text file.

- **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.

![Gradient options](image)

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 829
- “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.
6. Specify a point inside each area that you want to hatch and press Enter.
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.
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
Manages user coordinate systems.

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.

HPBACKGROUND COLOR
  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.

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 to display a preview when specifying internal points for a hatch.

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.

HSPACE
Sets the hatch pattern line spacing for user-defined patterns.

HPTRANSERCENCY
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 830

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
Manages user coordinate systems.

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 to display a preview when specifying internal points for a hatch.

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 789.

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.

See also:
- Scale Annotations on page 764
- Modify Hatch Alignment, Scale, and Rotation on page 830

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.

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 829
• Control How Overlapping Objects Are Displayed on page 383

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.
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.

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**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 831

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.
Press Enter twice to create the hatch.

Quick Reference

Commands
-HATCH
  conref to -Hatch blurb
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.
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 383
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 829 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 available from the ribbon or the Hatch Edit dialog box.

If the boundary object is a polyline or spline, *multi-functional grips* are displayed. For more information, see *Modify Objects with Multi-Functional Grips* on page 562.

**Modify the Extents of Nonassociative Hatches and Fills**

When you select a nonassociative 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.

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 nonassociative or an unbonded 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.

![Closed polyline created](image1) ![Wipeout object created from polyline](image2) ![Wipeout frame turned off](image3)

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 383

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

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 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.

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.

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.

Overview of Creating Text
The text you add to your drawings conveys a variety of information. It may be a complex specification, title block information, a label, or even part of the drawing.

Single-Line Text
For short entries that do not require multiple fonts or lines, create single-line text. Single-line text is most convenient for labels.

Multiline Text
For long, complex entries, create multiline, or paragraph text. Multiline text consists of any number of text lines or paragraphs that fit within a width you specify; it can extend vertically to an indefinite length.

Regardless of the number of lines, each set of paragraphs created in a single editing session forms a single object, which you can move, rotate, erase, copy, mirror, or scale.
There are more editing options for multiline text than there are for single-line text. For example, you can apply underlining, fonts, color, and text height changes to individual characters, words, or phrases within a paragraph.

**Annotative Text**

Use text for notes and labels in your drawing. You create annotative text by using an annotative text style, which sets the height of the text on the paper.

For more information about creating and working with an annotative text, see Create Annotative Text on page 775.

See also:

- Scale Annotations on page 764

**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.

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 at the Command prompt to enter the text at the Command prompt.

You can insert a field in single-line text. A field is text that is set up to display data that might change. When the field is updated, the latest value of the field is displayed.

The text styles used for single-line text are the same as those used for multiline text. When you create text, you assign an existing style by entering its name at the Style prompt. If you need to apply formatting to individual words and characters, use multiline text instead of single-line text.

You can also compress single-line text to fit between points that you specify. This option stretches or squeezes the text to fill the designated space.

The 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 892

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 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 MTEXT ribbon contextual tab (if the ribbon is active), or the In-Place Text Editor (or an alternative text editor, if the ribbon is not active) You can also use Command prompts. You can insert text from a file saved in ASCII or RTF format.

Before entering or importing text, you specify opposite corners of a text bounding box that defines the width of the paragraphs in the multiline text object. The length of the multiline text object depends on the amount of text, not the length of the bounding box. You can use grips to move or rotate a multiline text object.

**NOTE** Multiline text objects and imported text files are limited to 256 KB in size.

The MTEXT ribbon contextual tab and In-Place Text Editor display the bounding box with a ruler at the top. If the ribbon is not active, the Text Formatting toolbar is also displayed. The editor is transparent so that, as you create text, you can see whether the text overlaps other objects. To turn off transparency while you work, select Opaque Background on the Options menu. You can also make the background of the finished multiline text object opaque and set its color.

You can also insert fields in multiline text. A field is text that is set up to display data that might change. When the field is updated, the latest value of the field is displayed.

Text Style
Most characteristics of the text are controlled by the text style, which sets the default font and other options, such as line spacing, justification, and color. You can use the current text style or select a new one. The STANDARD text style is the default.
Within the multiline text object, you can override the current text style by applying formatting such as underlining, boldface, and different fonts to individual characters. You can also create stacked text, such as fractions or geometric tolerances and insert special characters, including Unicode characters, for TrueType fonts.

**NOTE** Not all SHX and TrueType text fonts support Unicode characters.

**Text Properties**

In the Properties palette, you can view and change the object properties of a multiline text object, including properties that apply specifically to text.

- **Justification** determines where text is inserted with respect to the bounding box and sets the direction of text flow as text is entered.
- **Line space options** control the amount of space between lines of text.
- **Width** defines the width of the bounding box and therefore controls where the text wraps to a new line.
- **Background** inserts an opaque background so that objects under the text are masked.

**To create multiline text**

1. Click Home tab ➤ Annotation panel ➤ Multiline Text.
2. Specify opposite corners of a bounding box to define the width of the multiline text object.
   - If the ribbon is active, the MTEXT ribbon contextual tab displays. If the ribbon is not active, the In-Place Text Editor is displayed.
3. To indent the first line of each paragraph, drag the first-line indent slider on the ruler. To indent the other lines of each paragraph, drag the paragraph slider.
4. To set tabs, click the ruler where you want a tab stop.
5. If you want to use a text style other than the default, on the ribbon, click the Annotate tab, Text panel. Select the desired text style from the drop-down list.
6. Enter text.
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 menu of options resets the character attributes of selected text to the current text style and also resets the text color to the color of the mtext object.

The text height setting specifies the height of capitalized text. For more information about how height is calculated, see MTEXT.

See also:

■ Use an Alternate Text Editor on page 929

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, and clear Allow Bullets and Lists.) When Allow Bullets and Lists is not checked, you cannot create new formatted lists in the multiline text object.
To create a list, use one of the following methods:

- Apply list formatting to new or selected text.
- Use Auto-list (on by default) and type the elements of a list.
- With Auto-list off, type the elements of a list and close and reopen the editor to convert the text to a list.

### Apply List Formatting

When you apply list formatting, you can specify bullets, uppercase or lowercase letters, or numbers. Default settings are used for the type of list you choose. Letters or numbers are followed by a period. Nested lists use a double bullet, letter, or number. Items are indented based on the tab stops on the ruler in the In-Place Text Editor.

### Use Auto-list to Type a List

When Auto-list is on, you can create a list as you type. You can use letters, numbers, or symbols.

For example, in the editor, enter \U+25CB, press TAB, and then enter some text. This creates a empty circle style bullet.

Not all symbols are available from the character map for a particular text font. However, if you specify the Unicode text directly (\U+25CB in this case), you can always get the bullet format of your choice.

**NOTE** Press TAB after you enter the Unicode text or symbol, or it will remain a separate character.

You can also paste a symbol from the Character Map dialog box.

The following characters can be used as punctuation after the number or letter when you type a list but cannot be used as bullets:

<table>
<thead>
<tr>
<th>Character</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>.</td>
<td>Period</td>
</tr>
<tr>
<td>:</td>
<td>Colon</td>
</tr>
<tr>
<td>)</td>
<td>Close parenthesis</td>
</tr>
<tr>
<td>&gt;</td>
<td>Close angle bracket</td>
</tr>
<tr>
<td>Character</td>
<td>Description</td>
</tr>
<tr>
<td>-----------</td>
<td>----------------------</td>
</tr>
<tr>
<td>]</td>
<td>Close square bracket</td>
</tr>
<tr>
<td>}</td>
<td>Close curly bracket</td>
</tr>
</tbody>
</table>

**Paste a List from Another Document**

If you copy a nested bulleted list (a list within a list) from Microsoft Word and paste the list into multiline text, the bullets that are displayed as empty circles cannot be formatted like other bullets in multiline text. This is because Word uses the letter o instead of a bullet for nested bulleted lists. You can remove formatting from the nested list and reapply to change the bullets to double bullets.

**To format multiline text as a list**

1. Click Home tab ➤ Annotation panel ➤ Multiline Text.
2. Specify opposite corners of a bounding box to define the width of the multiline text object.
3. To expand the Text Formatting toolbar, click Options button ➤ Show Options.
4. If you are converting multiline text to a list, select the paragraphs.

**NOTE** List formatting is only available when the Allow Bullets and Lists option is checked (the default).

5. On the expanded toolbar, click Numbering, Bullets, or Uppercase Letters.
   - **Numbering.** Uses numbers with periods for the items in a list.
   - **Bullets.** Uses a bullet or other character for the items in a list.
   - **Uppercase Letters.** Uses uppercase letters with periods for the items in a list. If the list has more items than the alphabet has letters, the sequence continues by using double letters. To use lowercase letters, right-click in the editor. Click Bullets and Lists ➤ Lettered ➤ Lowercase.

6. If you are creating new list items, enter the text.
7 To end the list, press ENTER to move to a new line. Click the button that you clicked to start the list.

8 To save your changes and exit the editor, use one of the following methods:
   - On the MTEXT ribbon contextual tab, in the Close panel, click Close Text Editor.
   - Click in the drawing outside the editor.
   - Press CTRL+ENTER.

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.

_Ceterum quidem omne spatium non ulta sed tempus est.
1. Urgent et circumstant utia undique nec resurgere aut.
2. Et immersos et in cupiditatem infixos premunt, numquam._

_Ceterum quidem omne spatium non ulta sed tempus est.
1. Urgent et circumstant utia.
2. Undique nec resurgere aut.
   Et immersos et
   Cupiditatem infixos
   Premunt, numquam._

**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 click the Stack button on the Text Formatting toolbar.

**Stack Numeric and Tolerance Characters Automatically**

You can specify that numeric characters entered before and after a slash, pound sign, or carat will stack automatically. For example, if you enter 1#3 followed by a nonnumeric character or space, the AutoStack Properties dialog box is displayed by default, and you can change the settings in the AutoStack dialog box to specify your formatting preferences.

The automatic stacking feature applies only to numeric characters immediately before and after the slash, pound sign, and carat. For tolerance stacking, the +, -, and decimal character also stack automatically.

**See also:**
- Use an Alternate Text Editor on page 929

**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.

   If you enter numbers separated by stack characters and then enter a nonnumeric character or press SPACEBAR, the AutoStack Properties dialog box is displayed.

   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 in either the MTEXT ribbon contextual tab or In-Place Text Editor and through the grip editing mode. Editing columns using grips allows you the flexibility of seeing the changes as you make them.

Columns follow a few rules. All columns have equal width and equal gutters. A gutter is the space between columns. The height of columns remains constant unless more text than the column can accommodate is added, or you manually move the editing grip to adjust the column height.

Editing Columns in the In-Place Text Editor

When you are working with columns using either the MTEXT ribbon contextual tab or In-Place Text Editor, the columns will be in a frame. If the Opaque background is turned on, the background covers each column, leaving gutter space blank. The ruler bar when applied, spans across all the columns, but the ruler is only active for the column that is selected as current.
Adding text to a column with an arbitrary height will not increase the column height even if text is already filling the column. Text will flow into another column.

You can also insert a column break to force text to start flowing into the next column. Anytime a column break is inserted, it is assumed that the current height of the column is fixed. To delete the break, highlight it and delete or use the Backspace key right after the break.

**Editing Columns in the Property Palette**

You will be able to select Static or Dynamic columns, turn off columns and change column and gutter width through the Property Palette. Changing column width in the palette will exhibit results similar to changing width using grips. The palette is the only place that you can also change gutter setting.

**To create multiple columns in the In-Place Text Editor**

1. If the ribbon is active, double-click a multiline text object to open the MTEXT ribbon contextual tab. If the ribbon is not active, the In-Place Text Editor is displayed.

2. In the In-Place Text Editor, select a column option and suboption from columns list. You have a choice between Dynamic or Static columns. You have two suboptions with Dynamic Columns – Auto height or Manual height. Static Columns allows you to choose the number of columns.

3. Adjust the column height by moving the arrows located on the bottom left of the first column.

**NOTE** The arrows on the ruler on the upper right side only adjust gutter width, not column width.
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 columns.

The following illustration demonstrates how grips are used with Dynamic Columns - Manual Height.

The following illustration demonstrates how grips are used with Static Columns.

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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 into your drawing by either importing the text or dragging a file icon from Windows Explorer.
You can insert TXT or RTF text files created in word processors into your drawing by either importing the text or dragging a file icon from Windows Explorer.

Importing TXT or RTF files from other sources gives you the most flexibility. For example, you can create a text file of standard notes that you include in drawings. The imported text becomes a multiline text object, which you can edit and reformat. Text imported from a TXT file inherits the current text style. Text imported from an RTF file inherits the current text style name, but retains its original fonts and format. Imported text files are limited to 256 KB and must have a file extension of .txt or .rtf.

If you drag a text file into a drawing, text width is determined by line breaks and carriage returns in the original document. When you drag an RTF file into a drawing, the text is inserted as an OLE object.

If you use the Clipboard to paste text from another application, the text becomes an OLE object. If you use the Clipboard to paste text from another file, the text is inserted as a block reference, and it retains its original text style.

See also:
- Import OLE Objects on page 1222

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.
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. Using the MLEADEREDIT command, you can add leaders to, or remove leaders from, an established multileader object.

Annotative multileaders containing multiple leader segments can have different head points in each scale representation. Horizontal landings and arrowheads can have different sizes, and landing gaps can have different distances, depending on the scale representation. The appearance of the horizontal landing within a multileader, as well as the type of leader line (straight or spline) and number of leader segments will remain the same in all scale representations. For more information, see Create Annotative Leaders and Multileaders on page 784.

You can use grips to modify the look of a multileader. Using grips, you can lengthen or shorten a landing or leader line, or move the entire leader object.
Arrange Leaders

Multileaders can be arranged to add order and consistency to your drawing. Multileader objects with blocks as content can be collected and attached to one landing line. Using the MLEADERCOLLECT command, multileaders can be collected horizontally, vertically, or within a specified area depending on your drawing needs.

Multileader objects can be sorted evenly along a specified line. Using the MLEADERALIGN command, selected multileaders can be aligned and evenly spaced as specified.

Associate Leaders with Objects

When associative dimensioning is turned on (DIMASSOC), the leader arrowhead can be associated with a location on an object using an object snap. If the object is relocated, the arrowhead remains attached to the object and the leader line stretches, but the multiline text remains in place.

See also:

- Create Annotative Leaders and Multileaders on page 784
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 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 On the Multileader toolbar, click Remove Leader.
3 Select the leader or 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.

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
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.

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.

Block placeholder fields can be used in block attribute definitions while you're working in the Block Editor.

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 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
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

System Variables

CTABLESTYLE
Sets the name of the current table style.

FIELDDISPLAY
Controls whether fields are displayed with a gray background.

FIELDVAL
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.

**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**

- DeviceName
- PaperSize
- PlotDate
- PlotOrientation
- PlotScale
- PageSetupName
- PlotStyleTable

For compatibility with previous versions, contextual fields in blocks and xrefs are not updated when you insert them in a drawing; instead, the field displays the last cached value. Therefore, if you want to use a contextual field within a block, for example, a title block, you must insert the field as an attribute.

**NOTE** The LispVariables and SheetSet Manager fields are not available in AutoCAD LT. The drawings created in AutoCAD that contain LispVariables or SheetSet Manager fields can be opened in AutoCAD LT and the cached value is displayed.

**List of contextual fields**

**Compatibility with Previous Releases**

When a drawing with fields is opened in AutoCAD 2004 or earlier, the fields are not updated; they display the value last displayed in the drawing before
it was opened. If no changes are made to a field, it is updated normally when it is reopened in a release that supports fields.

Fields are not available in the previous releases of AutoCAD LT. When a drawing with fields is opened in the previous releases of AutoCAD LT, the fields are evaluated based on the setting of the FIELDEVAL system variable in the drawing, but the FIELDEVAL system variable is not accessible.

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
UPDATEFIELD

System Variables
FIELDDISPLAY
Controls whether fields are displayed with a gray background.
FIELDVAL
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.

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
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

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 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.

<table>
<thead>
<tr>
<th>Formatting</th>
<th>Retained?</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bold</td>
<td>No</td>
</tr>
<tr>
<td>Color</td>
<td>Yes</td>
</tr>
<tr>
<td>Font</td>
<td>No</td>
</tr>
<tr>
<td>Height</td>
<td>No</td>
</tr>
<tr>
<td>Italic</td>
<td>No</td>
</tr>
<tr>
<td>Stacking</td>
<td>Yes</td>
</tr>
<tr>
<td>Underlining</td>
<td>Yes</td>
</tr>
</tbody>
</table>
Annotative Text Styles

Use text for notes and labels in your drawing. You create annotative text by using an annotative text style, which sets the height of the text on the paper. For more information about creating and working with an annotative text, see Create Annotative Text on page 775.

See also:
■ Scale Annotations on page 764

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 Microsoft Windows. Because SHX fonts are not recognized by Windows, a TrueType equivalent is supplied in the In-Place Text Editor when you select an SHX or any other non-TrueType font for editing.

See also:

Set Text Height on page 911

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 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

<table>
<thead>
<tr>
<th>Font File Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>@extfont2.shx</td>
<td>Japanese vertical font (a few characters are rotated to work correctly in vertical text)</td>
</tr>
<tr>
<td>bigfont.shx</td>
<td>Japanese font, subset of characters</td>
</tr>
<tr>
<td>chineset.shx</td>
<td>Traditional Chinese font</td>
</tr>
<tr>
<td>extfont.shx</td>
<td>Japanese extended font, level 1</td>
</tr>
<tr>
<td>extfont2.shx</td>
<td>Japanese extended font, level 2</td>
</tr>
<tr>
<td>gbcbig.shx</td>
<td>Simplified Chinese font</td>
</tr>
<tr>
<td>whgdtxt.shx</td>
<td>Korean font</td>
</tr>
<tr>
<td>whgtxt.shx</td>
<td>Korean font</td>
</tr>
<tr>
<td>whtgtxt.shx</td>
<td>Korean font</td>
</tr>
<tr>
<td>whmtxt.shx</td>
<td>Korean font</td>
</tr>
</tbody>
</table>

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

<table>
<thead>
<tr>
<th>Enter this ...</th>
<th>To specify this ...</th>
</tr>
</thead>
<tbody>
<tr>
<td>[font name],[big font name]</td>
<td>Both normal fonts and Big Fonts</td>
</tr>
<tr>
<td>[font name],</td>
<td>Only a normal font (Big Font unchanged)</td>
</tr>
</tbody>
</table>
Specifying fonts and Big Fonts at the Command prompt

<table>
<thead>
<tr>
<th>Enter this ...</th>
<th>To specify this ...</th>
</tr>
</thead>
<tbody>
<tr>
<td>[.big font name]</td>
<td>Only a Big Font (normal font unchanged)</td>
</tr>
<tr>
<td>[font name]</td>
<td>Only a normal font (Big Font, if any, removed)</td>
</tr>
<tr>
<td>ENTER (null response)</td>
<td>No change</td>
</tr>
</tbody>
</table>

**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 909

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 FONALT system variable. If you use a text style that uses a Big Font, you can map it to another font using the FONALT 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 906.

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 for AutoCAD, and acadlt.fmp for AutoCAD LT. You can change the font assignments in a font mapping file using any ASCII text editor.

For example, you could use the following entry in a font map file to specify that the timesnr.pfb font file be substituted with the times.ttf font file:

timesnr;times.ttf

The following table shows the font substitution rules used if a font file cannot be located when a drawing is opened.

<table>
<thead>
<tr>
<th>File extension</th>
<th>First mapping order</th>
<th>Second mapping order</th>
<th>Third mapping order</th>
<th>Fourth mapping order</th>
</tr>
</thead>
<tbody>
<tr>
<td>.ttf</td>
<td>Use font mapping table</td>
<td>Use font defined in text style</td>
<td>Windows substitutes a similar font</td>
<td></td>
</tr>
<tr>
<td>.shx</td>
<td>Use font mapping table</td>
<td>Use font defined in text style</td>
<td>Use FONTALT</td>
<td>Prompt for new font</td>
</tr>
<tr>
<td>.pfb</td>
<td>Use font mapping table</td>
<td>Use FONTALT</td>
<td>Prompt for new font</td>
<td></td>
</tr>
</tbody>
</table>

**Display Proxy Fonts**

For third-party or custom SHX fonts that have no TrueType equivalent, one of several different TrueType fonts called proxy fonts is substituted. In the In-Place Text Editor, proxy fonts look different from the fonts they represent to indicate that the proxy fonts are substitutions for the fonts used in the drawing.

Custom SHX fonts do not appear in the Font list on either the Text Editor Ribbon Contextual Tab or the Text Formatting toolbar. If you want to format characters by assigning one of these fonts, first create a text style that uses the font and then apply that text style to the characters.

**To specify a font mapping file**

1. Click Tools menu ➤ Options.
2 On the Files tab, in the list, double-click Text Editor, Dictionary, and Font File Names.

3 Double-click Font Mapping File.
   The acadlt.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.
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.

![Vertical Text Example](image)

**NOTE** Vertical orientation is not supported for TrueType fonts and symbols.

**Vertical Text for Asian Languages**

- **SHX fonts.** Text can be created with SHX fonts and Big Fonts for vertical display in the same way as for previous releases. For best results, use the single-line TEXT command, not MTEXT. You can select a vertical style in the Text Style dialog box.
TrueType fonts. You still select fonts starting with the @ sign, but now the text is automatically rotated 270 degrees. (In AutoCAD 2005 and earlier versions, you had to manually rotate this text.) Vertical cursor movement is now supported for vertical text.

To set vertical orientation in a text style

1. Click Home tab ➤ Annotation panel ➤ Text Style.
2. In the Text Style dialog box, select a text style from the Style Name list.
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. The procedures for modifying text vary slightly, depending on how the text was created.
Quick Reference

Commands
DDEDIT
Edits single-line text, dimension text, attribute definitions, and feature control frames.

FIND
Finds the text that you specify, and can optionally replace it with other text.

PROPERTIES
Controls properties of existing objects.

System Variables
MIRRTEXT
Controls how MIRROR reflects text.

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

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

- Text style assignment
- Justification
- Width
- Rotation
- Line spacing

In addition, you can use either the MTEXT ribbon contextual tab (if the ribbon is active) or the In-Place Text Editor (if the ribbon is not active) to modify individual formatting, such as boldface and underlining, and to change the width of the multiline text object.

Change Text Location

You can use grips to move multiline text or to resize the line width. A multiline text object has grips at the four corners of the text boundary and, in some cases, at the justification point.

If you use the Properties palette to move multiline text, you can edit content and change properties at the same time.
Commands such as DIMLINEAR or LEADER create multiline text automatically without requiring that a bounding box be specified; these objects have only a single grip at the justification point.

When you need to align or move multiline text objects, you can use the Node and Insertion object snaps for precision. If the OSNAPNODELEGACY system variable is set to 0, the Node object snap ignores multiline text.

See also:
- Work with Text Styles on page 901
- Control the Display of Polylines, Hatches, Gradient Fills, Lineweights, and Text on page 378

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.

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.

<table>
<thead>
<tr>
<th>Character</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td># (Pound)</td>
<td>Matches any numeric digit</td>
</tr>
<tr>
<td>@ (At)</td>
<td>Matches any alphabetic character</td>
</tr>
<tr>
<td>. (Period)</td>
<td>Matches any nonalphanumeric character</td>
</tr>
<tr>
<td>* (Asterisk)</td>
<td>Matches any string and can be used anywhere in the search string</td>
</tr>
<tr>
<td>? (Question mark)</td>
<td>Matches any single character; for example, ?BC matches ABC, 3BC, and so on</td>
</tr>
<tr>
<td>~ (Tilde)</td>
<td>Matches anything but the pattern; for example, ~AB* matches all strings that don't contain AB</td>
</tr>
<tr>
<td>[ ]</td>
<td>Matches any one of the characters enclosed; for example, [AB]C matches AC and BC</td>
</tr>
<tr>
<td>[-]</td>
<td>Matches any character not enclosed; for example, [-AB]C matches XC but not AC</td>
</tr>
<tr>
<td>[-]</td>
<td>Specifies a range for a single character; for example, [A-G]C matches AC, BC, and so on to GC, but not HC</td>
</tr>
<tr>
<td>` (Reverse quote)</td>
<td>Reads the next character literally; for example, `~AB matches ~AB</td>
</tr>
</tbody>
</table>

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 51.

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.
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.
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.

<table>
<thead>
<tr>
<th>Format code</th>
<th>Purpose</th>
<th>Enter this …</th>
<th>To produce this …</th>
</tr>
</thead>
<tbody>
<tr>
<td>\0…\o</td>
<td>Turns overline on and off</td>
<td>Autodesk \OAutoCAD\o</td>
<td></td>
</tr>
<tr>
<td>\L…\l</td>
<td>Turns underline on and off</td>
<td>Autodesk \LAutoCAD\l</td>
<td></td>
</tr>
<tr>
<td>~</td>
<td>Inserts a nonbreaking space</td>
<td>Autodesk AutoCAD~LT</td>
<td></td>
</tr>
<tr>
<td>\</td>
<td>Inserts a backslash</td>
<td>Autodesk \AutoCAD</td>
<td></td>
</tr>
<tr>
<td>{…}</td>
<td>Inserts an opening and closing brace</td>
<td>Autodesk {AutoCAD LT}</td>
<td></td>
</tr>
<tr>
<td>\Cvalue;</td>
<td>Changes to the specified color</td>
<td>Autodesk \C2;AutoCAD</td>
<td></td>
</tr>
<tr>
<td>\filename;</td>
<td>Changes to the specified font file</td>
<td>Autodesk \Ftimes; AutoCAD</td>
<td></td>
</tr>
<tr>
<td>Format code</td>
<td>Purpose</td>
<td>Enter this …</td>
<td>To produce this …</td>
</tr>
<tr>
<td>-------------------</td>
<td>--------------------------------------------------------------------------</td>
<td>------------------------</td>
<td>-------------------</td>
</tr>
<tr>
<td>\H{value};</td>
<td>Changes to the text height specified in drawing units</td>
<td>Autodesk \H2;AutoCAD</td>
<td></td>
</tr>
<tr>
<td>\H{value};</td>
<td>Changes the text height to a multiple of the current text height</td>
<td>Autodesk \H3x;AutoCAD</td>
<td></td>
</tr>
<tr>
<td>\S...^...;</td>
<td>Stacks the subsequent text at the /, #, or ^ symbol</td>
<td>1.000\S+0.010^-0.000;</td>
<td></td>
</tr>
<tr>
<td>\T{value};</td>
<td>Adjusts the space between characters. Valid values range from a minimum of .75 to 4 times the original spacing between characters.</td>
<td>\T2;Autodesk</td>
<td></td>
</tr>
<tr>
<td>\Q{angle};</td>
<td>Changes obliquing angle</td>
<td>\Q20;Autodesk</td>
<td></td>
</tr>
<tr>
<td>\W{value};</td>
<td>Changes width factor to produce wide text</td>
<td>\W2;Autodesk</td>
<td></td>
</tr>
<tr>
<td>\A</td>
<td>Sets the alignment value; valid values: 0, 1, 2 (bottom, center, top)</td>
<td>\A1;\S1/2</td>
<td></td>
</tr>
<tr>
<td>\P</td>
<td>Ends paragraph</td>
<td>Autodesk\PAutoCAD</td>
<td></td>
</tr>
</tbody>
</table>

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.

**Big text**

over text/ under text

Baseline: 1 \1/2
Center: 1 \1/2
Topline: 1 \1/2

Tolerances: 1.0000.010
Architectural: 9-11/16"n

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

A table is a rectangular array of cells that contain annotation, primarily text but also blocks. Tables appear in many different forms on many of the sheets that make up drawing sets. In the AEC industry, tables are often referred to as “schedules” and contain information about the materials needed for the construction of the building being designed. In the manufacturing industry, they are often referred to as “BOM” (bills of materials).

The table object creates a table of any size that can be used for any purpose, including as a list or index to a set of drawing sheets to be published.

Create and Modify Tables

A table is an object that contains data in rows and columns. A table object can be created from an empty table or table style. A table can also be linked to data in a Microsoft Excel spreadsheet.

After the table has been created, you can click any gridline on the table to select it and then modify it by using the Properties palette or grips.
When you change the height or width of the table, only the row on page 1406 or column on page 1392 adjacent to the grip you have selected will change. The table will maintain its height or width. To change the size of the table proportionally to the size of the row or column you are editing, press Ctrl while using a column grip.

**Break Tables into Multiple Parts**

A table with a large amount of data can be broken into primary and secondary table fragments. Use the table breaking grips found at the bottom of your table to make a table span multiple columns in your drawing or to manipulate the different table parts you have already created.

**Modify a Table Cell**

You can click inside a cell to select it. Grips are displayed in the middle of the cell borders. Click inside another cell to move selection to that cell. Drag the grips on a cell to make the cell and its column or row larger or smaller.

**NOTE** When a cell is selected, press F2 to edit the cell text.
To select more than one cell, click and drag over several cells. You can also hold down Shift and click inside another cell to select those two cells and all the cells between them.

When you click inside a table cell when the ribbon is active, the Table ribbon contextual tab is displayed. If the ribbon is not active, the Table toolbar is displayed. From here, you can

- Edit rows and columns
- Merge and unmerge cells
- Alter the appearance of cell borders
- Edit data formatting and alignment
- Lock and unlock cells from editing
- Insert blocks, fields, and formulas
- Insert blocks 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 945

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
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 amultiline text object.
SPELL
   Checks spelling in a drawing.
TABLE
Creates an empty table object.

TABLEEDIT
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

CTABLESTYLE
Sets the name of the current table style.

FIELDDISPLAY
Controls whether fields are displayed with a gray background.

FIELDVAL
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 DATALINKUPDATE command. 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.

**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 901

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.

**TABLEEDIT**
- 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
CTABLESTYLE
Sets the name of the current table style.
FIELDDISPLAY
Controls whether fields are displayed with a gray background.
FIELDVAL
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, the Text Formatting toolbar is displayed, and you can begin entering text. The row height of the cell increases to accommodate the number of lines of text. To move to the next cell, press TAB, or use the arrow keys to move left, right, up, and down. You can quickly edit cell text by pressing F2 in a selected cell.
When you insert a block into a table cell, either the block can be automatically fit to the size of the cell, or the cell can adjust to accommodate the size of the block. Blocks can be inserted from the Table toolbar, or from the shortcut menu. Multiple blocks can be inserted in a table cell. If there is more than one block in a table cell, use the Manage Cell Content dialog box to customize the way the cell content is displayed.
Inside the cell, the arrow keys move the cursor. Use the Table toolbar and the shortcut menu to format text, import text, or make other changes to the text in the cell.

See also:

- Create Multiline Text on page 848

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.
2. 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.

Quick Reference

FIELD
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.
Use Formulas in Table Cells

Table cells can contain formulas that do calculations using the values in other table cells.

With a table cell selected, you can insert formulas from the Table toolbar as well as the shortcut menu. You can also open the In-Place Text Editor and enter a formula in a table cell manually.

Insert a Formula

In formulas, cells are referred to by their column letter and row number. For example, the cell at top left in the table is A1. Merged cells use the number of what would be the top-left cell. A range of cells is defined by the first and last cells, with a colon between them. For example, the range A5:C10 includes cells in rows 5 through 10 in columns A, B, and C.

A formula must start with an equal sign (=). The formulas for sum, average, and count ignore empty cells and cells that do not resolve to a numeric value. Other formulas display an error (#) if any cell in the arithmetic expression is empty or contains nonnumeric data.
Use the Cell option on the shortcut menu to select a cell in another table in the same drawing. When you have selected the cell, the In-Place Text Editor opens so you can enter the rest of the formula. You can also insert a formula using the Table toolbar.

**Copy a Formula**

When you copy a formula to another cell in the table, the range changes to reflect the new location. For example, if the formula in A10 sums A1 through A9, when you copy it to B10, the range of cells changes so that it sums B1 through B9.

If you don’t want a cell address to change when you copy and paste the formula, add a dollar sign ($) to the column or row part of the address. For example, if you enter $A10, the column stays the same and the row changes. If you enter $A$10, 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.

**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.
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 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
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
CTABLESTYLE
   Sets the name of the current table style.
FIELDDISPLAY
   Controls whether fields are displayed with a gray background.
FIELDVAL

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

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.

DIMORIGINATE
 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.

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.

■ **Nonassociative dimensions.** Selected and modified with the geometry they measure. Nonassociative dimensions do not change when the geometric objects they measure are modified. The dimension variable DIMASSOC is set to 1.

■ **Exploded dimensions.** Contain a collection of separate objects rather than a single dimension object. The DIMASSOC system variable is set to 0.

You can determine whether a dimension is associative or nonassociative by selecting the dimension and doing one of the following:

■ Use the Properties palette to display the properties of the dimension.

■ Use the LIST command to display the properties of the dimension.

You can also use the Quick Select dialog box to filter the selection of associative or nonassociative dimensions. A dimension is considered associative even if only one end of the dimension is associated with a geometric object. The DIMREASSOCIATE command displays the associative and nonassociative elements of a dimension.
Special Situations and Limitations

You may need to use DIMREGEN to update associative dimensions after panning or zooming with a wheel mouse, after opening a drawing that was modified with an earlier release, or after opening a drawing with external references that have been modified.

Although associative dimensions support most object types that you would expect to dimension, they do not support the following:

- Hatches
- 2D solids
- Objects with nonzero thickness
- Images
- DWF underlays

When selecting objects to dimension, make sure that the objects that you select don't include a directly overlapping object that does not support associative dimensioning such as a 2D solid.

Associativity is not maintained between a dimension and a block reference if the block is redefined.

Associativity is not maintained between a dimension and a 3D solid if the shape of the 3D solid is modified.

For information about working with associative dimensions in combination with previous releases, see Save Drawings to Previous Drawing File Formats on page 1301.

See also:

- Change Dimension Associativity on page 1047

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

```
1  15   15  2
first extension line suppressed
```

■ Specify how far beyond from the dimension line the extension line extends (overshoot)

```
0.75
extension line overshoot
```

■ Control the extension origin offset, the distance between the extension line origin, and the start of the extension line
Specify a fixed length for extension lines, as measured from the dimension line toward the extension line origin.

Specify a noncontinuous linetype, typically used for centerlines.

Modify the angle of the extension lines of a selected dimension to make them oblique.

**Fixed-Length Extension Lines**

With the Dimension Style Manager, on the Lines tab, you can specify a dimension style that sets the total length of extension lines starting from the dimension line toward the dimension origin point.
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 1009

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 LT 2006, the arrowhead directions will revert to their original orientations.

See also:

- Customize Arrowheads on page 972

**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 669.

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 \textit{arrowhead size overall scale}. The dimension line is trimmed by \textit{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 671.

If you use paper-space scaling, the scale factor is computed before applying it to the arrowhead size value.

To use your own arrowhead symbol

1 Click Home tab ➤ Annotation panel ➤ Dimension Style.
2 In the Dimension Style Manager, select the style you want to change. Click Modify.
3 In the Dimension Style Manager, Symbols and Arrows tab, under Arrowheads, select User Arrow from the First arrowhead list.
4 In the Select Custom Arrow Block dialog box, enter the name of your block. Click OK.
5 To choose a different custom arrowhead for the second arrowhead, repeat steps 3 and 4, choosing User Arrow from the Second arrowhead list (optional).
6 Click OK.
7 Click Close to exit the Dimension Style Manager.

Quick Reference

Commands

BLOCK

Creates a block definition from selected objects.

DIMSTYLE

Creates and modifies dimension styles.

WBLOCX

Writes objects or a block to a new drawing file.
**System Variables**

**DIMASZ**
Controls the size of dimension line and leader line arrowheads.

**DIMBLK**
Sets the arrowhead block displayed at the ends of dimension lines.

**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 Dimension Style Manager, Fit tab. For example, you can specify that text and arrowheads be kept together. In this case, if there is not room for both between the extension lines, they are both placed outside. You can specify that if there is room for only text or arrowheads, then either text only or arrowheads only are placed between the extension lines.

The following illustrations show how the program applies a "best fit" for arrowheads and text.

If there is no room for text between the extension lines, you can have a leader line created automatically. This is useful in cases where text outside the extension lines would interfere with other geometry, for example, in continued dimensions. Whether text is drawn to the right or the left of the leader is controlled by the horizontal justification setting on the Text tab of the Modify/New Dimension Style dialog box. Also, you can fit text and arrowheads by changing their size.

Even if the arrowheads are outside the extension lines, you can have a line drawn between the extension lines. This is called forcing an internal line and is illustrated as follows.
Fit Diameter Dimension Text

You can draw several different diameter dimensions depending on text placement, horizontal settings on the Text tab, and whether you select the Draw Dim Line Between Ext Lines option on the Fit tab.

To place text within extension lines

1. Click Home tab ➤ Annotation panel ➤ Dimension Style.
2. In the Dimension Style Manager, select the style you want to change. Click Modify.
3. In the Modify Dimension Style dialog box, Fit tab, under Fit Options, select an option.
4. Click OK.
5. Click Close to exit the Dimension Style Manager.
If there is enough room, text is fit between extension lines.

**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.
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.

The default alignment is horizontal dimension text, even for vertical dimensions.

**Position Dimension Text Horizontally**

The position of the text along the dimension line in relation to the extension lines is referred to as text placement. To place text yourself when you create a dimension, use the Place Text Manually When Dimensioning option on the Fit tab of the Modify/New Dimension Style dialog box. Use the text placement options to automatically place text at the center of the dimension line, at either extension line, or over either extension line.
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.

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.
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.
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.
DIMTXXDIRECTION
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 Dimension Style Manager, Text tab. You can choose a text style while creating a dimension style and specify a text color and a height independent of the current text style's height setting. You can also specify the gap between base dimension text and the box that surrounds it.

The text styles used for dimensions are the same text styles used by all text created in your drawing.

For more information, see Work with Text Styles on page 901.
Supply User Text to Dimensions

In addition to the prefixes and suffixes specified for primary and alternate units, you can supply your own text as you create a dimension. Because the prefix, suffix, and user-supplied text form a single text string, you can represent tolerance stacks and apply changes to font, text size, and other characteristics using the text editor.

To add user text above and below the dimension line, use the separator symbol \X. Text that precedes this symbol is aligned with and above the dimension line. Text that follows the \X symbol is aligned with and below the dimension line. The space between the dimension line and the text is determined by the value you enter under Gap in the Annotation dialog box.

Example: User Text in Dimensions

In this example, the primary dimension measurement is 5.08, and the alternate dimension measurement is 2.00. The primary units have the suffix H7/h6, and the alternate units have the suffix inches.

At the text prompt, while creating the dimension, you enter the following format string:

<> H7/h6\XSee Note 26\P[ ]

The angle brackets represent the primary units, and the square brackets represent the alternate units. The \X separates text above the dimension line from text below the dimension line. The \P is a paragraph break.

The resulting text appears as follows:

To control the text style in dimensions

1. Click Home tab ➤ Annotation panel ➤ Dimension Style.
2. In the Dimension Style Manager, select the style you want to change. Click Modify.
3. In the Modify Dimension Style dialog box, Text tab, under Text Appearance, select a text style.
4 If the current text style does not have a fixed height, enter the height of
dimension text in the Text Height box.

5 Under Tolerances, enter a height for tolerance values in the Scaling for
Height box.

6 In the Offset from Dim Line box, enter a value for the gap around base
dimension text.

7 Select a color from the Text Color box.

8 Click OK

9 Click Close to exit the Dimension Style Manager.

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

System Variables

DIMCLR

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 (Diameter (unicode 2205 and R, respectively).

These settings are available on the Primary Units tab of the Dimension Style Manager.
Control the Display of Alternate Units

You can create dimensions in two systems of measurement simultaneously. A common use of this feature is to add feet and inches dimensions to drawings created using metric units. The alternate units appear in square brackets ([ ]) in the dimension text. Alternate units cannot be applied to angular dimensions.

If alternate-units dimensioning is on when you edit a linear dimension, the measurement is multiplied by an alternate scale value that you specify. This value represents the number of alternate units per current unit of measurement. The default value for imperial units is 25.4, which is the number of millimeters per inch. The default value for metric units is about 0.0394, which is the number of inches per millimeter. The number of decimal places is specified by the precision value for alternate units.

For example, for imperial units, if the alternate scale setting is the default value, 25.4, and the alternate precision is 0.00, the dimension might look like the following figure.

To add and format primary units

1. Click Home tab ➤ Annotation panel ➤ Dimension Style.
2. In the Dimension Style Manager, select the style you want to change. Click Modify.
3. In the Modify Dimension Style dialog box, Primary Units tab, under Linear or Angular Dimensions, select a unit format and precision value for the primary units.
4. Under Linear Dimensions, enter any prefix and suffix for the displayed dimension.
5. Click OK.
6. Click Close to exit the Dimension Style Manager.
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.

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 0.0000 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

<table>
<thead>
<tr>
<th>Option</th>
<th>Effect</th>
<th>Examples</th>
</tr>
</thead>
<tbody>
<tr>
<td>No options selected</td>
<td>Includes zero feet and zero inches</td>
<td>0'-0 1/2&quot; 0'-6&quot; 1'-0&quot; 1'-0 3/4&quot;</td>
</tr>
<tr>
<td>0 Inches selected</td>
<td>Suppresses zero inches (includes zero feet)</td>
<td>0'-0 1/2&quot; 0'-6&quot; 1&quot; 1'-0 3/4&quot;</td>
</tr>
<tr>
<td>0 Feet selected</td>
<td>Suppresses zero feet (includes zero inches)</td>
<td>1/2&quot; 6&quot; 1'-0&quot; 1'-0 3/4&quot;</td>
</tr>
<tr>
<td>0 Feet and 0 Inches selected</td>
<td>Suppresses zero feet and zero inches</td>
<td>1/2&quot; 6&quot; 1&quot; 1'-0 3/4&quot;</td>
</tr>
</tbody>
</table>

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.
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 1050

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 Primary Units tab of the Dimension Style Manager.
To specify the fraction format

1  Click Home tab ➤ Annotation panel ➤ Dimension Style.
2  In the Dimension Style Manager, select the style you want to change. Click Modify.
3  In the Modify Dimension Style dialog box, Primary Units tab, under Linear dimensions, select one of the following from Fraction format:
   ■ Horizontal
   ■ Diagonal
   ■ Not Stacked
4  Click OK.
5  Click Close to exit the Dimension Style Manager.

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.
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.

See also:
- Draw, Scale, and Annotate in Model Space on page 257
- Scale Views in Layout Viewports on page 284
- Scale Annotations on page 764
The DMLFAC 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.
DMLFAC
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.

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.
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 multilime 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.

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 Modify Dimension Style dialog box, Symbols and Arrows tab, under Center Marks. You can also access this setting with the DIMCEN system variable.
The size of the centerline is the length of the centerline segment that extends outside the circle or arc. It is also the size of the gap between the center mark and the start of the centerline. The size of the center mark is the distance from the center of the circle or arc to the end of the center mark.

Create Jogged Radius Dimensions

With the DIMJOGGED command, you can create jogged radius dimensions, also called “foreshortened radius dimensions,” when the center of an arc or circle is located off the layout and cannot be displayed in its true location. The origin point of the dimension can be specified at a more convenient location called the center location override.

You can control the default angle of the jog in the Modify Dimension Style dialog box, Symbols and Arrows tab, under Radius Dimension Jog.
Once a jogged radius dimension is created, you can modify the jog and the center location override by

- Using grips to move the features
- Changing the locations of the features with the Properties palette
- Using STRETCH

**NOTE** Jogged radius dimensions can be viewed but not edited in versions previous to AutoCAD 2006. Also, if you make dramatic changes to the associated geometry, you may get unpredictable results for the jogged radius dimension.

See also:

- Fit Dimension Text Within Extension Lines on page 974

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.

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 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 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.

![Diagram showing feature location and leader endpoint]

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.

UCS
Manages user coordinate systems.

Create Arc Length Dimensions

Arc length dimensions measure the distance along an arc or polyline arc segment.

Typical uses of arc length dimensions include measuring the travel distance around a cam or indicating the length of a cable. To differentiate them from linear or angular dimensions, arc length dimensions display an arc symbol by default.

The arc symbol, also called a hat or cap, is displayed either above the dimension text or preceding the dimension text. You can specify the placement style using the Dimension Style Manager. The placement style can be changed on the Symbols and Arrows tab of either the New Dimension Style dialog box or the Modify Dimension Style dialog box.

The extension lines of an arc length dimension can be orthogonal or radial.
NOTE Orthogonal extension lines are displayed only when the included angle of the arc is less than 90 degrees.

To create an arc length dimension

1. Click Home tab ➤ Annotation panel ➤ Arc Length.
2. Select an arc or polyline arc segment.
3. Specify the dimension line location.

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 adjust the placement of linear dimensions so they are evenly spaced.

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.

Inspection Dimension

Inspection dimensions allow you to effectively communicate how frequently manufactured parts should be checked to ensure that the dimension value and tolerances of the parts are within the specified range.

When working with parts that need to meet 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.

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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.

**Dimension Breaks**

With dimension breaks, you can keep the dimension, extension, or leader lines from appearing as if they are a part of the design.

Dimension breaks can be added to a dimension or a multileader automatically or manually. The method that you choose to place dimension breaks depends on the number of objects that intersect a dimension or multileader.
You can add dimension breaks to the following dimension and leader objects:

- Linear dimensions (aligned and rotated)
- Angular dimensions (2- and 3-point)
- Radial dimensions (radius, diameter, and jogged)
- Arc length dimensions
- Ordinate dimensions
- Multileaders (straight only)

The following dimension and leader objects do not support dimension breaks:

- Multileaders (spline only)
- “Legacy” leaders (straight or spline)

The following table explains the conditions where dimension breaks do not work or are not supported.

<table>
<thead>
<tr>
<th>Condition</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>No break in xrefs or blocks</td>
<td>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.</td>
</tr>
<tr>
<td>No break on arrowhead and dimension text</td>
<td>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 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.</td>
</tr>
<tr>
<td>No break on trans-spatial dimensions</td>
<td>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.</td>
</tr>
</tbody>
</table>
You can move dimension breaks from dimensions or multileaders. When removing dimension breaks from a dimension or multileader, all dimension breaks are removed. If there are some dimension breaks that you don’t want to remove, you need to add them again.

The following objects can be used as cutting edges when adding a dimension break:

- Dimension
- Leader
- Line
- Circle
- Arc
- Spline
- Ellipse
- Polyline
- Text
- Multiline text
- Blocks but limited to the previously mentioned objects in this list
- Xrefs but limited to the previously mentioned objects in this list

**Automatic Dimension Breaks**

To create an automatically placed dimension break, you select a dimension or multileader, and then use the Auto option of the DIMBREAK command. Automatic dimension breaks are updated any time the dimension or multileader, or intersecting objects are modified.

You control the size of automatically placed dimension breaks on the Symbols and Arrows tab of the Dimension Style dialog box. The specified size is affected by the dimension break size, dimension scale, and current annotation scale for the current viewport. For more information about annotation scaling, see *Scale Annotations* on page 764.

**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.

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 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.

Modify Dimension Text

Once you've created a dimension, you can change the location and orientation of the existing dimension text or replace it with new text.

Once you've created a dimension, you can rotate the existing text or replace it with new text. You can move the text to a new location or back to its home position, which is the position defined by the current dimension style. In the following illustration, the home position is above and centered on the dimension line.
When you rotate or replace dimension text, you specify the change first, for example, rotating the text to be at an angle. When you move dimension text, you select a single dimension to move.

You can move dimension text to the left, right, or center along the dimension line or to any position inside or outside the extension lines. A quick and simple way to do this is by using grips. If you move text up or down, the current vertical alignment of the text relative to the dimension line is not changed, so the dimension and extension lines are modified accordingly. The following illustration shows the result of moving text down and to the right. The text remains centered vertically in relation to the dimension line.

See also:
- Control Dimension Text on page 974

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 Click Dimension menu ➤ Align Text ➤ Home.
2 Select the dimension text you want to return to its home position.

**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.
**Modify Dimension Geometry**

Grip editing is the quickest and easiest way to modify the location of dimension elements. How you edit dimensions depends whether the dimension is associative.

You can modify dimensions with the editing commands and with grip editing. Grip editing is the quickest and easiest way to modify dimensions. How you edit dimensions depends on whether the dimension is associative.

**Modify Associative Dimensions**

Associative dimensions retain their associativity to dimensioned objects through many editing commands if both the dimension and the associated geometry are selected and operated on with a single command. For example, if a dimension and its associated geometry are moved, copied, or arrayed in
the same command, each dimension retains associativity with its respective geometry.

In some circumstances, dimensions are automatically disassociated, including

- If the associated geometric object is erased
- If the associated geometric object undergoes a Boolean operation such as UNION or SUBTRACT
- If grip editing is used to stretch a dimension parallel to its dimension line
- If the association to a geometric object is specified using the Apparent Intersection object snap, and the geometric object is moved so that the apparent intersection no longer exists

In other circumstances, a dimension may become partially associated. For example, if a linear dimension is associated with the endpoints of two geometric objects and one of the objects is erased, the remaining association is preserved. The disassociated end of the linear dimension may then be associated with another geometric object using DIMREASSOCIATE.

NOTE The Command prompt displays a warning message if a dimension is disassociated.

Modify Nonassociative Dimensions

For nonassociative dimensions, when you edit dimensioned objects, you must include the relevant dimension definition points in the selection set, or the dimension is not updated. Definition points determine the dimension location. For example, to stretch a dimension, you must include the appropriate definition points in the selection set. You can easily include them by turning on grips and selecting the object so that the grips are highlighted.

The definition points for each type of dimension are indicated in the following illustrations. The middle point of the dimension text is a definition point for all dimension types.
If no angle vertex is shown, definition points are placed at the ends of the lines that form the angle. In the two-line angular example, a definition point is placed at the center point of the dimensioned arc.

**NOTE** Definition points are drawn on a special layer named DEFPOINTS, which is not plotted.

**Modify Exploded Dimensions**

You can edit exploded dimensions as you would any other objects because an exploded dimension is a collection of separate objects: lines, 2D solids, and text. Occasionally you may need to explode a dimension to make changes such as creating a break in a dimension line or extension line. Once a dimension is exploded, you cannot reassociate the dimension into a dimension object.
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.

See also:

- Control Dimension Geometry on page 965
Add associativity to dimensions in legacy drawings.

Remove associativity from dimensions in drawings that will be used by people working in releases prior to AutoCAD 2002, but who do not want any proxy objects in the drawings.

Reassociate Dimensions to Different Objects

With DIMREASSOCIATE, you can select one or more dimensions and step through the extension-line origin points of each dimension. For each extension-line origin point, you can specify a new association point on a geometric object. Association points determine the attachment of extension lines to locations on geometric objects.

NOTE When you create or modify associative dimensions, it is important to locate their association points carefully so that if you make a future design change, the geometric objects that you change will also change the dimensions associated with them.

When you use the DIMREASSOCIATE command, a marker is displayed that indicates whether each successive extension line origin point of the dimension is associative or nonassociative. A square with an X in it means that the point is associated with a location on an object, while an X without the square means that the point is not associated with an object. Use an object snap to specify the new association for the extension-line origin point or press Enter to skip to the next extension-line origin point.

NOTE The marker disappears if you pan or zoom with a wheel mouse.

Change Nonassociative Dimensions to Associative

You can change all the nonassociative dimensions in a drawing to associative. Use QSELECT to select all nonassociative dimensions, and then use DIMREASSOCIATE to step through the dimensions, associating each one with locations on geometric objects.

Change Associative Dimensions to Nonassociative

You can change all associative dimensions in a drawing to nonassociative dimensions. Use QSELECT to select all associative dimensions, and then use DIMDISASSOCIATE to convert them into nonassociative dimensions.

See also:

- Associative Dimensions on page 959
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.

Add Geometric Tolerances

You can add geometric tolerances that show acceptable deviations of form, profile, orientation, location, and runout of a feature.

Overview of Geometric Tolerances

Geometric tolerances show acceptable deviations of form, profile, orientation, location, and runout of a feature.

You add geometric tolerances in feature control frames. These frames contain all the tolerance information for a single dimension. Geometric tolerances can be created with or without leader lines, depending on whether you create them with TOLERANCE or LEADER.

A feature control frame consists of two or more components. The first feature control frame contains a symbol that represents the geometric characteristic to which a tolerance is being applied, for example, location, profile, form, orientation, or runout. Form tolerances control straightness, flatness, circularity and cylindricity; profiles control line and surface. In the illustration, the characteristic is position.
You can use most editing commands to change feature control frames, and you can snap to them using the object snap modes. You can also edit them with grips.

**NOTE** Unlike dimensions and leaders, geometric tolerances cannot be associated with geometric objects.

You can also create tolerances. For more information about creating and working with an annotative tolerances, see Create Annotative Dimensions and Tolerances on page 779.

**See also:**
- Scale Annotations on page 764

**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.
Plot and Publish Drawings
You prepare your drawing for plotting or publishing by specifying page setup settings. These settings are stored in the drawing file with the layout. Once a layout is established, you can modify the settings of its page setup or apply a different page setup.

Quick Start to Saving Settings for Plotting and Publishing

Preparing a drawing for plotting or publishing requires specifying many settings and options that define the output of your drawing. To save time, you can save these settings as a named page setup.

You can apply a named page setup to paper space layouts using the Page Setup Manager. You can also import a named page setup from another drawing and apply it to layouts in the current drawing.

Quick Reference

Commands

PAGESETUP
Controls the page layout, plotting device, paper size, and other settings for each new layout.

PSETUPIN
Imports a user-defined page setup into a new drawing layout.
Specify Page Setup Settings

Page setups are associated with layouts and stored in the drawing file. The settings specified in a page setup determine the appearance and format of your final output.

Overview of Page Setup Settings

A page setup is a collection of plot device and other settings that affect the appearance and format of your final output. These settings can be modified and applied to other layouts.

After you complete a drawing on the Model tab, you can begin creating a layout to plot by clicking a layout tab. When you click a layout tab for the first time, a single viewport is displayed on the page. A dashed line indicates the printable area of the paper for the currently configured paper size and plotter.

Once you have your layout set up, you specify the settings for the layout's page setup, which includes the plot device settings and other settings that affect the appearance and format of the output. The settings you specify in the page setup are stored in the drawing file with the layout. You can modify the settings of a page setup at any time.
By default, every initialized layout has a page setup associated with it. You can initialize a layout by clicking on its tab to activate the previously unused layout. A layout does not contain any plot settings before initialization. A layout must be initialized (its paper size can be defined in the page setup to any size other than 0 x 0) before it can be published. Once initialized, layouts can be drawn upon, published, and added to sheet sets as sheets (after the drawing has been saved). You can apply a named page setup saved with one layout to another layout. This creates a new page setup with the same settings as the first one.

If you want the Page Setup Manager to be displayed each time you begin a new drawing layout, select the Show Page Setup Manager for New Layouts option on the Display tab in the Options dialog box. If you don't want a viewport to be automatically created for each new layout, clear the Create Viewport in New Layouts option on the Display tab in the Options dialog box.

To modify the settings of a layout's page setup

1. Click the layout tab for which you want to modify the page setup settings.

2. Click Output tab ➤ Plot panel ➤ Page Setup Manager.

3. In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.

4. Click Modify.

5. In the Page Setup dialog box, modify the required settings. Click OK.

6. In the Page Setup Manager, click Close.

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

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

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 1094
■ “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.

Select a Paper Size for a Layout

You can select a paper size from a standard list, or you can add custom paper sizes using the Plotter Configuration Editor.

You can select a paper size from a standard list. The paper sizes available in the list are determined by the plot device that is currently selected for the layout. If your plotter is configured for raster output, you must specify the output size in pixels. You can add custom paper sizes, which are stored in the plotter configuration (PC3) file, using the Plotter Configuration Editor.

If you are using a system printer, the paper size is determined by the document defaults that are set in the Windows Control Panel. The default paper size is
displayed in the Page Setup dialog box when you create a new layout for that configured device. If you change the paper size in the Page Setup dialog box, the new paper size is saved with the layout and overrides the size saved in the plotter configuration (PC3) file.

See also:

■ “Control PC3 File Device and Document Settings” in the Driver and Peripheral Guide

To set the paper size for a layout

1. Click the layout tab for which you want to set the paper size.

2. Click Output tab ➤ Plot panel ➤ Page Setup Manager.

3. In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.

4. Click Modify.

5. In the Page Setup dialog box, under Paper Size, select a paper size from the list. Click OK.

6. In the Page Setup Manager, click Close.

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.

**PLOTTERMANAGER**

Displays the Plotter Manager, where you can add or edit a plotter configuration.
Set the Plot Area of a Layout

You can specify the plot area to determine what will be included in the plot.

When you prepare to plot from the Model tab or a layout tab, you can specify
the plot area to determine what will be included in the plot. When you create
a new layout, the default Plot Area option is Layout. Layout plots all objects
within the printable area of the specified paper size.

The Display Plot Area option plots all the objects displayed in the drawing.
The Extents Plot Area option plots all the visible objects in the drawing. The
View Plot Area option plots a saved view. You can use the Window Plot Area
option to define an area to be plotted.

See also:
- Specify the Area to Plot on page 1095

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.

**Adjust the Plot Offset of a Layout**

The printable area of a drawing sheet is defined by the selected output device and is represented by the dashed line in a layout. When you change the output device, the printable area may change.

The plot offset specifies an offset of the plot area relative to the lower-left corner (the origin) of the printable area or the edge of the paper, depending on the Specify Plot Offset Relative To option specified in the Options dialog box, Plot and Publish tab. The Plot Offset area of the Plot dialog box displays the specified plot offset option in parentheses.

You can offset the geometry on the paper by entering a positive or negative value in the X and Y Offset boxes. However, this may result in the plot area being clipped.

If you choose to plot an area other than the entire layout, you can also center the plot on the sheet of paper.
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.

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 284
- Draw, Scale, and Annotate in Model Space on page 257

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**

SCALELISTEDIT

Controls the list of scales available for layout viewports, page layouts, and plotting.

PAGESETUP

Controls the page layout, plotting device, paper size, and other settings for each new layout.

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.
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.

Select a Plot Style Table for a Layout

A plot style table is a collection of plot styles assigned to a layout or the Model tab.

A plot style is an object property, similar to linetype and color. 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 1101

**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.
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.

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, or Hidden option.

Shaded viewport plotting options apply to all objects in viewports and model space. If you use the Shaded option, plot style tables included in the page setup do not affect plots.

NOTE  Shaded viewport plotting requires a raster-capable device. Most modern plotters and printers are raster-capable devices.

Plot Options

The following options that can be specified for layouts affect how objects are plotted.

■ Plot Object Lineweights. Specifies that lineweights assigned to objects and layers are plotted.

■ Plot with Transparency. Specifies that transparency applied to objects and layers is plotted.
- **Plot with Plot Styles.** Specifies that the drawing is plotted using plot styles. Selecting this option automatically plots lineweights. If you do not select this option, objects are plotted with their assigned properties and not with the plot style overrides.

- **Plot Paper Space Last.** Specifies that objects in model space are plotted before those in paper space.

- **Hide Paperspace Objects.** Specifies whether the Hide operation applies to objects in the paper space viewport. This option is available only from a layout tab. The effect of this setting is reflected in the plot preview, but not in the layout.

**See also:**
- **Set Options for Plotted Objects** on page 1104

**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.

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.

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. 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 tab or current layout.

Create and Use Named Page Setups

You can save plot device and other page setup settings as named page setups that can be modified and imported into other drawings.

You can create named page setups and apply them to other layouts in your drawing. Named page setups are saved in the drawing file and can be imported into other drawing files and applied to other layouts. You can also modify the settings of named page setups. If you modify a named page setup, you can choose whether the modifications apply to the current layout or to all the layouts in the current drawing that use the named page setup.

If you want to plot the same layout more than one way, or if you want to specify the same output options for several layouts, use named page setups.
You can apply different named page setups to the same layout to achieve specific results when plotting. For example, you might create the named page setups in the following table to control scaling and paper size.

<table>
<thead>
<tr>
<th>Page setup name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NoScaling</td>
<td>Plot at scale 1:1, E-size sheet</td>
</tr>
<tr>
<td>Scale 1 to 2</td>
<td>Plot at scale 1:2, C-size sheet</td>
</tr>
<tr>
<td>Draft</td>
<td>Plot to the draft-quality plotter</td>
</tr>
<tr>
<td>Final</td>
<td>Plot to the high-quality plotter</td>
</tr>
<tr>
<td>Fit-to-Paper</td>
<td>Fit to Paper, A-size sheet</td>
</tr>
</tbody>
</table>

Once you specify a named page setup for a layout, whenever you plot or publish the layout, it is plotted or published with the settings specified in the named page setup set for the layout.

To create a new named page setup

1. Click Output tab ➤ Plot panel ➤ Page Setup Manager.
2. In the Page Setup Manager, Page Setups area, click New.
3. In the New Page Setup dialog box, enter a name for the new page setup.
4. Under Start With, select a page setup in the list. The settings specified in the selected page setup will be displayed in the Page Setup dialog box after you click OK.
5. Click OK.
6. In the Page Setup dialog box, change any required settings. Click OK. The new page setup is displayed in the Page Setups list in the Page Setup Manager.
7. To apply the new page setup to the current layout, in the Page Setup Manager, click Set Current.
8. In the Page Setup Manager, click Close.
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.
Once you have completed a drawing, you can use a number of methods to output the drawing. You can plot the drawing on paper or create a file for use with another application. In either case, you select the plot settings.

**Quick Start to Plotting**

To print a single layout or part of a drawing, use the Plot dialog box.

Use a named page setup or change the settings in the Plot dialog box to define the output of your drawing.

To output more than one drawing, use the Publish dialog box.

**Quick Reference**

**Commands**

PLOT

Plots a drawing to a plotter, printer, or file.

**Overview of Plotting**

Understanding terms and concepts that relate to plotting makes your first plotting experience in the program easier.
**Plotter Manager**

The Plotter Manager is a window that lists plotter configuration (PC3) files for every non-system printer that you install. Plotter configuration files can also be created for Windows® system printers if you want to use default properties different from those used by Windows. Plotter configuration settings specify port information, raster and vector graphics quality, paper sizes, and custom properties that depend on the plotter type.

The Plotter Manager contains the Add-a-Plotter wizard, which is the primary tool for creating plotter configurations. The Add-a-Plotter wizard prompts you for information about the plotter you want to set up.

**Layouts**

A layout represents a plotted page. You can create as many layouts as you need. Each layout is saved on its own layout tab and can be associated with a different page setup.

Elements that appear only on a plotted page, such as title blocks and notes, are drawn in paper space in a layout. The objects in the drawing are created in model space on the Model tab. To view these objects in the layout, you create layout viewports.

**Layout Initialization**

Layout initialization is a process in which a previously unused layout is made active, by clicking on its tab.

A layout does not contain any plot settings before initialization. Once initialized, layouts can be drawn upon, published, and added to sheet sets as sheets (after the drawing has been saved).

**Page Setups**

When you create a layout, you specify a plotter and settings such as page size and plot orientation. These settings are saved in a page setup. You can control these settings for layouts and for the Model tab using the Page Setup Manager. You can name and save page setups for use with other layouts.

If you don’t specify all the settings in the Page Setup dialog box when you create a layout, you can set up the page just before you plot. Or you can override a page setup at plot time. You can use the new page setup temporarily for the current plot, or you can save the new page setup.
Plot Styles

A plot style controls how an object or layer is plotted by determining plotted properties such as lineweight, color, and fill style. Plot style tables collect groups of plot styles. The Plot Style Manager is a window that shows all the plot style tables available.

There are two plot style types: color-dependent and named. A drawing can use only one type of plot style table. You can convert a plot style table from one type to the other. You can also change the type of plot style table a drawing uses once it has been set.

For **color-dependent plot style tables**, an object’s color determines how it is plotted. These plot style table files have `.ctb` extensions. You cannot assign color-dependent plot styles directly to objects. Instead, to control how an object is plotted, you change its color. For example, all objects assigned the color red in a drawing are plotted the same way.

**Named plot style tables** use plot styles that are assigned directly to objects and layers. These plot style table files have `.stb` extensions. Using them enables each object in a drawing to be plotted differently, independent of its color.

Plot Stamps

A plot stamp is a line of text that is added to your plot. You can specify where this text is located on the plot in the Plot Stamp dialog box. Turn this option on to add specified plot stamp information—including drawing name, layout name, date and time, and so on—to a drawing that is plotted to any device. You can choose to record the plot stamp information to a log file instead of plotting it, or in addition to plotting it.

**NOTE** A drawing file or drawing template file that was created with an educational version will always be plotted with the following plot stamp: PRODUCED BY AN AUTODESK EDUCATIONAL PRODUCT. Blocks and xrefs created with an educational version and used in a commercial version will also result in the educational plot stamp being plotted.

See also:

- Create Multiple-View Drawing Layouts (Paper Space) on page 263
- “To view the custom properties Help” in the *Driver and Peripheral Guide*
To plot a drawing

1  Click Output tab ➤ Plot panel ➤ Plot.
2  In the Plot dialog box, under Printer/Plotter, select a plotter from the Name list.
3  Under Paper Size, select a paper size from the Paper Size box.
4  (Optional) Under Number of Copies, enter the number of copies to plot.
5  Under Plot Area, specify the portion of your drawing to plot.
6  Under Plot Scale, select a scale from the Scale box.
7  For more options, click the More Options button.

8  (Optional) Under Plot Style Table (Pen Assignments), select a plot style table from the Name box.
9  (Optional) Under Shaded Viewport Options and Plot Options, select any appropriate settings.

NOTE Plot stamping happens at plot time and is not saved with the drawing.

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 tab or current layout.
PLOT
Plots a drawing to a plotter, printer, or file.

PLOTSTAMP
Places a plot stamp on a specified corner of each drawing and logs it to a file.

PLOTTERMANAGER
Displays the Plotter Manager, where you can add or edit a plotter configuration.

STYLESMANAGER
Displays the Plot Style Manager, where you can revise plot style tables.

VIEWPLOTDETAILS
Displays information about completed plot and publish jobs.

System Variables

BACKGROUND PLOT
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.

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.
Select a Printer or Plotter

Before plotting a drawing, you must select a printer or plotter. The device you select affects the printable area of the drawing.

After selecting a printing or plotting device, you also can easily plot a drawing using the default settings in the Plot dialog box.

To select a printer or plotter

1. Click Output tab ➤ Plot panel ➤ Plot.
2. In the Plot dialog box, under Printer/Plotter, select a plotter from the Name list.
   If you have already selected a paper size and it is not supported by the plotter you have chosen, you are warned that a paper size supported by the plotter will be used. Click OK if the warning is displayed.
3. Once you have selected a plotter, you can continue to select a paper size, or if the paper size is correct, click OK to plot the drawing.

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
In the Plot dialog box, select the paper size that you want to use.

If you plot from a layout, you may have already specified a paper size in the Page Setup dialog box. However, if you plot from the Model tab, you need to specify a paper size when you plot. In the Plot dialog box, select the paper size that you want to use. The list of paper sizes depends on the printer or plotter that you have selected in either the Plot or Page Setup dialog box. The list of available plotters includes all those that are currently configured for use with Windows and those for which you have installed non-system drivers.

You can also set the default page size used to create new layouts for most plotters by editing the PC3 file associated with that plotter. For Windows system printers, you can use this technique to specify different default page sizes for Windows and for this program.

NOTE If the PAPERUPDATE system variable is set to 0, you are prompted if the layout’s existing paper size is not supported by the plotter you have selected. If the PAPERUPDATE system variable is set to 1, the paper size is automatically updated to reflect the default paper size of the selected plotter.

Use a Custom Paper Size
If you need to specify a paper size that is not listed in either the Plot dialog box or the Page Setup dialog box, you can add a custom paper size for a non-system plotter using the Plotter Configuration Editor. Typically, you cannot add a custom paper size to Windows system printers because the allowable page sizes and printable areas are determined by the manufacturer. However, you can modify the printable area for paper sizes associated with a Windows system printer.

To select a paper size for the current plot

1. Click Output tab ➤ Plot panel ➤ Plot.
2. In the Plot dialog box, under Printer/Plotter, select a plotter in the Name box.
3. Under Paper Size, select a paper size from the list.
The paper sizes that are listed depend on the plotter you selected.

**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.
   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 drawing orientation.

Specify the Printable Area
The printable area is displayed by a dashed border in a layout. The plotter and paper size you select determine the printable area.

**WARNING** If you set your plotter to use paper-saving features such as plotting inked area or nesting, your plotter will probably not use the printable area and plot offset specifications.

If your plotter reports an incorrect printable area for your paper size, you can adjust the printable area in the Modify Standard Paper Sizes area under the Modify Standard Paper Sizes (Printable Area) option on the Device and Document Settings tab in the Plotter Configuration Editor.

**NOTE** The Modify Standard Paper Sizes option is not a margins feature. Specify where your drawing is plotted on the page in the Plot Offset area in the Plot dialog box.

Quick Reference

Commands
PAGESETUP
Controls the page layout, plotting device, paper size, and other settings for each new layout.
PLOT
Plots a drawing to a plotter, printer, or file.

PLOTTERMANAGER
Displays the Plotter Manager, where you can add or edit a plotter configuration.

Set the Position of the Plot
The printable area of a drawing sheet is defined by the selected output device and is represented by the dashed line in a layout. You can change the position of plot relative to the printable area or to the edge of the paper.

The printable area of a drawing sheet is defined by the selected output device and is represented by the dashed line in a layout. When you change to another output device, the printable area may change.

The settings in the Plot Offset area of the Plot dialog box specify an offset of the plot area relative to the lower-left corner (the origin) of the printable area or the edge of the paper, depending on the setting made in the Specify Plot Offset Relative To option (Options dialog box, Plot and Publish tab). The Plot Offset area of the Plot dialog box displays the specified plot offset option in parentheses.

You can offset the drawing on the paper by entering positive or negative values in the X and Y offset boxes. However, this may result in the plot area being clipped. If the Plot Area is not set to Layout (Extents, Display, View, or Window), you can also select the Center the Plot option.

Quick Reference

Commands

PAGESETUP
Controls the page layout, plotting device, paper size, and other settings for each new layout.

PLOT
Plots a drawing to a plotter, printer, or file.

PLOTTERMANAGER
Displays the Plotter Manager, where you can add or edit a plotter configuration.
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 the Model tab or a layout:

- On the Model tab, you can establish the scale in the Plot dialog box. This
  scale represents a ratio of plotted units to the world-size units you used to
draw the model.

- In a layout, you work with two scales. The first affects the overall layout
  of the drawing, which usually is scaled 1:1, based on the paper size. The
  second is the scale of the model itself, which is displayed in layout
  viewports. The scale in each of these viewports represents a ratio of the
  paper size to the size of the model in the viewport.

NOTE You can modify the list of scales that are displayed in all view and plot scale
lists with SCALELISTEDIT.

Set a Specific Scale

When you plot, the paper size you select determines the unit type, inches or
millimeters. For example, if the paper size is in mm, entering 1 under mm
and 10 under Units produces a plotted drawing in which each plotted
millimeter represents 10 actual millimeters.

The illustrations show a light bulb plotted at three different scales.

Scale the Drawing to Fit the Page

When you review drafts, a precise scale is not always important. You can use
the Fit to Paper option to plot the view at the largest possible size that fits the
paper. The height or width of the drawing is fit to the corresponding height
or width of the paper.
When you 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 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. By default, the system variable honors the setting in the Page Setup or the Plot dialog boxes.

■ **Plot with Plot Styles.** Specifies that the drawing is plotted using plot styles. Selecting this option automatically plots lineweights. If you do not select this option, objects are plotted with their assigned properties and not with the plot style overrides.
- **Plot Paper Space Last.** Specifies that objects in model space are plotted before those in paper space.

- **Hide Paperspace Objects.** Specifies whether the Hide operation applies to objects in the paper space viewport. This option is available only from a layout tab. The effect of this setting is reflected in the plot preview, but not in the layout.

- **Plot Stamp On.** Turns on plot stamping and places a plot stamp on a specified corner of each drawing and/or logs it to a file. Plot stamp settings are specified in the Plot Stamp dialog box, where you can specify the information you want applied to the plot stamp, such as drawing name, date and time, plot scale, and so on. To open the Plot Stamp dialog box, select Plot Stamp On in the Plot dialog box, and then click the Plot Stamp Settings button.

- **Save Changes to Layout.** Saves changes you make in the Plot dialog box to the layout if you click OK.

**To set shaded plotting options from the Model tab**

1. Click Output tab ➤ Plot panel ➤ Plot.
2. In the Plot dialog box, Shaded Viewport Options area, Shade Plot drop-down list, select an option.

**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](image1) ![Hidden lines removed](image2)

**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](image3) ![Hidden lines removed](image4)

**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 controls an object's plotted properties.
A plot style is an object property, similar to linetype and color. A plot style can be assigned to an object or assigned to a layer. A plot style controls an object's plotted properties, including

- Color
- Dither
- Grayscale
- Pen number
- Virtual pen
- Screening
- Linetype
- Lineweight
- 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.

**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.

DEFPSTYLED

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.

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.

   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 a configuration file (CFG) or plotter configuration file (PCP or PC2) to import pen settings, base the new plot style table on an existing plot style table, or start from scratch. If you use an existing plot style table, the new plot style table uses the same type of table as the original. Click Next.
4. On the Pick Plot Style Table page, select Color-Dependent Plot Style Table or Named Plot Style Table.
5. If you are importing pen settings from a PCP, PC2, or CFG file, or if you are basing the new plot style table on an existing plot style table, specify the file on the Browse File Name page. If you use a CFG file, you may need to select which plotter's configuration to import. Click Next.
6. On the File Name page, enter a name for the new plot style table. Click Next.
7. On the Finish page, you can edit the new plot style table by choosing Plot Style Table Editor. You can assign the new plot style table so that it can be used in all drawings.
8  Click Finish.
The new plot style table is available in both the Plot and the Page Setup
dialog boxes for all drawings using color-dependent plot style tables.

**To rename a plot style table**

1  Click Output tab ➤ Plot panel ➤ Plot Style Manager.
2  Right-click the plot style whose name you want to change. Click Rename.
3  Enter the new file name. Be sure to add the same file extension (.ctb or .stb). When finished, press ENTER.
The renamed plot style table is available in the Plot and Page Setup dialog boxes for all drawings using that type of plot style table.

**To change a plot style table description**

1  Click Output tab ➤ Plot panel ➤ Plot Style Manager.
2  Double-click the plot style table file whose description you want to change.
3  In the Plot Style Table Editor, General tab, enter the new description for the plot style table.
4  Click Save & Close.

**To edit plot styles in a plot style table**

1  Click Output tab ➤ Plot panel ➤ Plot Style Manager.
2  Double-click the plot style table you want to change.
3  In the Plot Style Table Editor, Form View tab, under Plot Styles, select a plot style and edit the settings.
4  Click Save & Close.
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>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>acadlt.ctb</td>
<td>Default plot style table</td>
</tr>
<tr>
<td>fillPatterns.ctb</td>
<td>Sets first 9 colors to use first 9 fill patterns, all others to use object's fill</td>
</tr>
<tr>
<td>grayscale.ctb</td>
<td>Converts all colors to grayscale when plotted</td>
</tr>
<tr>
<td>monochrome.ctb</td>
<td>Plots all colors as black</td>
</tr>
<tr>
<td>None</td>
<td>Applies no plot style table</td>
</tr>
<tr>
<td>screening 100%.ctb</td>
<td>Uses 100% ink for all colors</td>
</tr>
<tr>
<td>screening 75%.ctb</td>
<td>Uses 75% ink for all colors</td>
</tr>
</tbody>
</table>

Use Color-Dependent Plot Style Tables | 1117
### Table

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>screening 50%.ctb</code></td>
<td>Uses 50% ink for all colors</td>
</tr>
<tr>
<td><code>screening 25%.ctb</code></td>
<td>Uses 25% ink for all colors</td>
</tr>
</tbody>
</table>

**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 1113

## 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 View 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 1125

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.

- **acadlt.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 1113

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 Output tab ➤ Plot panel ➤ Plot Style Manager.
2. Double-click the plot style table that you want to modify.
3. In the Plot Style Table Editor, General tab, click Delete AutoCAD Release 14 Color Mapping Table.
4. Read the warning. Click Yes to delete the color mapping table, or click No to keep it.
5. Click Save & Close.

**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 Output tab ➤ Plot panel ➤ Plot Style Manager.
2. Double-click the plot style table you want to modify.
3. In the Plot Style Table Editor, Form View tab, under Plot Styles, select the plot style you want to modify.
4. Under Properties, click the arrow next to the property you want to change. Select an option from the list.
5. Edit other properties or plot styles as needed.
6. Click Save & Close.

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 Output tab ➤ Plot panel ➤ Plot Style Manager.
2. Right-click a CTB or STB file. Click Open.
3. In the Plot Style Table Editor, Table View tab, click the Color field for the plot style you want to change.
4. On the Color drop-down list, click the color you want to use or click Select Color to display the Select Color dialog box and do one of the following:
   - On the Index tab, click a color or enter the ACI color number (1-255) or name in the Color box. Click OK.
   - On the True Color tab, in the Color Model box, specify a color. (Enter a color value in the Color box or specify values in the Hue, Saturation, and Luminance boxes.) Click OK.
   - On the Color Books tab, in the Color Book box, select a color (use the up and down arrow and click on a color chip.) Click OK.

To use screening

1. Click Output tab ➤ Plot panel ➤ Plot Style Manager.
2. Double-click the plot style table that you want to modify.
3. In the Plot Style Table Editor, Form View tab, select the plot style you want to change.
4. In the Screening box, enter an intensity value between 1 and 100.
5. When finished, click Save & Close.

   You can edit properties for multiple plot styles while in the Plot Style Table Editor.

**NOTE** A common practice when using screening is to set the plotted color to black for each style so that the screening will be a percentage of black.
To enable or disable dithering

1. Click Output tab ➤ Plot panel ➤ Plot Style Manager.
2. Double-click the plot style table that you want to modify.
3. In the Plot Style Table Editor, Form View tab, select the plot style you want to change and select Dither.
4. Select On or Off.
5. When finished, click Save & Close.
   You can edit properties for multiple plot styles while in the Plot Style Table Editor.

To enable or disable conversion to grayscale

1. Click Output tab ➤ Plot panel ➤ Plot Style Manager.
2. Double-click the plot style table that you want to modify.
3. In the Plot Style Table Editor, Form View tab, select the plot style you want to change and select Grayscale.
4. Select On or Off.
5. When finished, click Save & Close.
   You can edit properties for multiple plot styles while in the Plot Style Table Editor.

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 Output tab ➤ Plot panel ➤ Plot Style Manager.
2. Double-click the plot style table that you want to modify.
3. In the Plot Style Table Editor, Form View tab, select the plot style you want to change and click Virtual Pen.
4. Under Virtual Pen #, enter a number between 1 and 255 or enter 0 or **Automatic** to have the program assign the ACI color of the object you are plotting to the virtual pen.
5. When finished, click Save & Close.

You can edit properties for multiple plot styles while in the Plot Style Table Editor.
Quick Reference

Commands

STYLES MANAGER
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 359
■ Control Lineweights on page 370
To set the plotted lineweight

1. Click Output tab ➤ Plot panel ➤ Plot Style Manager.
2. Double-click the plot style table that you want to modify.
3. In the Plot Style Table Editor, Form View tab, select the plot style you want to change.
4. Click the Lineweight arrow and select a lineweight from the list. To specify that the object's lineweight should be used select Use Object Lineweight.
5. Click Save & Close.
   You can edit properties for multiple plot styles while in the Plot Style Table Editor.

To set the plotted linetype

1. Click Output tab ➤ Plot panel ➤ Plot Style Manager.
2. Double-click the plot style table that you want to modify.
3. In the Plot Style Table Editor, Form View tab, select the plot style you want to change.
4. Click the Linetype arrow and select a linetype from the list. To specify that the object's linetype should be used, select Use Object Linetype.
5. To adjust the linetype scale to show the complete pattern, on the Form View tab, select On in the Adaptive box.
6. Click Save & Close.
   You can edit properties for multiple plot styles while in the Plot Style Table Editor.

To apply scaling to non-ISO linetypes and fill patterns

1. Click Output tab ➤ Plot panel ➤ Plot Style Manager.
2. Double-click the plot style table that you want to modify.
3 In the Plot Style Table Editor, General tab, select Apply Global Scale Factor to Non-ISO Linetypes.
   This option scales linetypes and fill patterns according to a value that you specify.
4 In the Scale Factor box, enter a scale factor to apply.
5 Click Save & Close.

Quick Reference

Commands

STYLES MANAGER
   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
The default setting for Line Join Style is Use Object Join Style, which is rounded. Assign a line join style in a plot style to override the object’s default line join style at plot time.

**To assign a line end or line join style**

1. Click Output tab ➤ Plot panel ➤ Plot Style Manager.
2. Right-click a CTB or STB file. Click Open.
3. In the Plot Style Table Editor, Table View tab, click the Line End Style or Line Join Style field for the plot style you want to change.
4. Select an option from the drop-down list.

**Quick Reference**

**Commands**

STYLES MANAGER

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
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 829

To assign a fill style

1. Click Output tab ➤ Plot panel ➤ Plot Style Manager.
2. Right-click a CTB or STB file. Click Open.
3. In the Plot Style Table Editor, Table View tab, click the Fill Style field for the plot style you want to change.
4. Select a fill style from the drop-down list.

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 other plot style options.

When you preview your drawing, the active toolbars and tool palettes are hidden and a temporary Preview toolbar is displayed that provides buttons to plot, pan, and zoom the drawing.

In the Plot and Page Setup dialog boxes, a thumbnail preview is also displayed, which shows the printable area and the position of the drawing on the page.

To preview a plot

1. Click Output tab ➤ Plot panel ➤ Plot.
2. In the Plot dialog box, click Preview.
   A preview window opens, and the cursor changes to the real-time zoom cursor.
3. Right-click to display a shortcut menu with the following options: Plot, Pan, Zoom, Zoom Window, or Zoom Original (to zoom to the original preview magnification).
4. Press ESC to exit the preview and return to the Plot dialog box.
5. If necessary, make additional adjustments to the plot settings and preview the plotted drawing again.
6. Once the settings are correct, click OK to plot the drawing.

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.

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 1149
- Review and Markup Files with Design Review on page 1363
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 1149
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 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.
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
- “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

Publishing provides a streamlined alternative to plotting multiple drawings. You can easily publish a set of drawings 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.

Publishing provides an easy way to create an electronic drawing set. 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, 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 either 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, DWFx, or PDF file
Multiple single-sheet DWF, DWFx, or PDF files

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.

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

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.
BACKGROUND PLOT
Controls whether background plotting is turned on or off for plotting and publishing.

Create and Modify a Drawing Set for Publishing

You can assemble a collection of drawing sheets to publish all at once to single or multiple DWF, DWFx, or PDF files. 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 directly to an electronic format that can be distributed using E-mail, FTP sites, project web sites, or CD. You can save a drawing set that’s 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, either in the shortcut menu or in the Publish dialog box, all the layouts in the current drawing are listed in the sheetlist.

2. In the Publish dialog box, you can modify the list of sheets by doing any of the following as needed:
   - **Add sheets.** To add sheets from other drawings, click the Add Sheets button (or drag drawings from the desktop). In the Select Drawings dialog box, select drawings. Click Select to add them to the list of sheets in the Publish dialog box. All of the layouts in a drawing become individual sheets in the list of drawing sheets. You can remove the individual sheets if you do not want them to become part of the drawing set. A layout must be initialized (its paper size must be defined in the page setup to any size other than 0 x 0) before it can be published.
NOTE To include all layouts when you add sheets to a drawing set, ensure the Include Layouts When Adding Sheets option is selected either in the shortcut menu or in the Publish dialog box.

- **Include model layouts.** If you include an uninitialized (the paper size is not defined in the page setup or is set to 0 x 0) model layout, it will be marked as Uninitialized in the Status column on the sheet list. It can be plotted if you select an override page setup for it in the Publish dialog box from the page setup drop-down list under Page Setups in the sheet list.

NOTE To include the model when you add sheets to a drawing set, ensure the Include Model When Adding Sheets option is selected either in the shortcut menu or in the Publish dialog box.

- **Remove sheets.** To remove sheets from the list, select one or more sheets in the list. Click the Remove Sheets button. To remove all sheets, right-click the selection. Click Remove All.

- **Reorder sheets.** To reorder the sheets up or down one position in the list, select a sheet. Click either the Move Sheet Up or Move Sheet Down button. Sheets in the drawing set are viewed or plotted in the order shown in the list.

- **Rename sheets.** To rename a sheet, select it in the list and right-click. Click Rename Sheet. Enter the new sheet name.

- **Change page setups.** To change the page setup for a layout, select the sheet, and in the Page Setup list select a named page setup from the list; or select one or more sheets and right-click. Click Change Page Setup. In the Page Setup list, select a page setup, or select Import to import page setups from another drawing or template. In the Import Page Setups dialog box, select a drawing with one or more page setups. Click Import. In the Page Setup list, select a named page setup.

NOTE Change the page setup for each layout depending on your desired output. Model space page setups can only be applied to model space sheets; paper space page setups can only be applied to paper space sheets.

- **Copy sheets.** To copy one or more drawing sheets, select the sheets in the list and right-click. Click Copy Selected Sheets. The copied drawing sheets are added and highlighted at the end of the sheet list. When a sheet is copied, its name is created by the addition of -Copy(n)
at the end of the original sheet name. For example, if you create one copy of a sheet called Plumbing, the copied sheet is called Plumbing-Copy(1). Each time you copy the same sheet, the $n$ is incremented by 1. By creating copies of a sheet, you can have different page setups and other settings for the same sheet.

3 When your list of drawing sheets is assembled and configured the way you want for your drawing set, click the Save List button.

4 In the Save List As dialog box, in the File Name box, enter a name for the list. Click Save.
   The drawing set list is saved as a DSD (Drawing Set Descriptions) file.

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**

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.

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.

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 1164
- Review and Markup Files with Design Review on page 1363

**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
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.

PUBLISHHATCH
Controls whether hatch patterns published to DWF or DWFx format are treated as a single object when they are opened in Autodesk Impression.

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. You can easily republish a collection of drawing sheets if you’ve saved a description of the drawing set as a DSD (Drawing Set Descriptions) file.

To republish a drawing set

1 Click Output tab ➤ Publish panel ➤ Publish.
2 If you have a drawing open, in the Publish Drawing Sheets dialog box, right-click. Click Remove All to delete the layouts from the list of drawing sheets.
3 In the Publish dialog box, click the Load Sheet List button.
4 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 list.
5. In the Publish To group, click to select the DWF Format. Click DWF File or DWFx File.

6. Click Publish to start the process.
   The animated plotter icon on the right side of the status bar indicates that the publish job is in progress.

7. 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 is 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, 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 on Autodesk Design Review, see Review and Markup Files with Design Review on page 1363.
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 1363

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:

- 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. Click Publish Options.
3 In the Publish Options dialog box, change the desired settings. Click OK.

**NOTE** DWF or DWFx passwords are case-sensitive. The password or phrase can consist of letters, numbers, punctuation marks, or non-ASCII characters. Keep a list of passwords and their corresponding DWF or DWFx file names in a safe place. If you lose or forget the password, it cannot be recovered. PDF files cannot be password protected.

### To be prompted for a name for a multi-sheet DWF, DWFx, or PDF file

1 Click Output tab ➤ Plot panel ➤ Batch Plot.
2 In the Publish To group, click DWF Format.
3 Click DWF File or DWFx File.
4 Click Publish Options.
   The Publish Options dialog box appears.
5 In the General DWF Options pane, under DWF Type, select Multi-sheet DWF from the drop-down list.
6 In the Multi-sheet DWF Options, select Prompt for Name to be prompted for a file name every time you publish a DWF file.
7 Click OK.
8 In the Publish dialog box, continue 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 In the Publish To group, click DWF Format.
3 Click DWF File or DWFx File.
4 Click Publish Options.
   The Publish Options dialog box appears.
5 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.

6 Click OK.

7 In the Publish dialog box, continue 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 In the Publish To group, click DWF Format.

3 Click DWF File or DWFx File.

4 Click Publish Options. The Publish Options dialog box appears.

5 In the DWF Data Options pane, under Block Information, click to display the drop-down list, and select Include.

6 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.

7 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.

8 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.

9 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.

10 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.

11 In the Publish dialog box, continue 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 In the Publish To group, click DWF Format.
3 Click DWF File or DWFx File.

4 Click Publish Options.
   The Publish Options dialog box appears.

5 In the DWF Data Options pane, under Block Information, click to display
   the drop-down list, and select Include.

6 Under DWF Data Options, Block Template File, click to display the
   drop-down list, and select Edit.

7 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.

8 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.

9 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.

10 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.

11 Click Save. Click OK.
   The modified block template file is now available under DWF Data
   Options in the Publish Block Template dialog box.

12 In the Publish dialog box, continue 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.
Share Data Between Files
Reference Other Drawing Files

You can attach an entire drawing to the current drawing as a referenced drawing. With xrefs, changes made in the referenced drawing are reflected in the current drawing.

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.

Overview of Referenced Drawings (Xrefs)

You can attach an entire drawing file to the current drawing as a referenced drawing (xref). With xrefs, changes made in the referenced drawing are reflected in the current drawing. 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.

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 88

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

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.

VISRETAINT

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.
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, 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.

**Control the Properties of Xref’s 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 1184
- Clip External References and Blocks on page 1190
- Set Paths to Referenced Drawings on page 1181
- DesignCenter
- Set Interface Options on page 107

**To attach an xref**

1. Click Blocks & References tab ➤ Reference panel ➤ DWG.
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.

VISRETAI
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.

XREFNNOTIFY
Controls the notification for updated or missing xrefs.

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. This is the most specific but least flexible option.

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
- 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.

**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.

Quick Reference

Commands
XREF
Starts the EXTERNALREFERENCES command.

System Variables

Detach Referenced Drawings
To completely remove DWG references (xrefs) from your drawing, you need to detach them rather than erase them.

To completely remove xrefs from your drawing, you need to detach 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 Bind Referenced Drawings
Beyond attaching and detaching referenced drawings (xrefs), you can also update and bind drawing references.

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.

NOTE You can only constrain objects in the drawing file to the insertion point on an Xref, and not its nested objects.

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.

![Diagram showing xref and overlay relationships between drawings]

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.

To overlay an xref

1. Click Blocks & References tab ➤ Reference panel ➤ DWG.
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.

Update Referenced Drawing Attachments

When you open a drawing, all DWG references (xrefs) update automatically. You can also update xrefs whenever you want to ensure that the most current versions are displayed in your drawing.

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 1212.

### 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.
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 1207.

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 1308.

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$n$.
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
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 1283
- Clip Underlays

To clip an xref

1. Click Blocks and References 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.
Set XCLIPFRAME, DWFFRAME, DGNFRAME, IMAGEFRAME, or PDFFRAME to 1. This specifies that the clipping boundary is displayed.

Select the clipped boundary to see the grip at the midpoint on the first edge of the clipped boundary.

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.

DWFCCLIP
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.

IMAGEFRAME
Controls whether image frames are displayed and plotted.
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**

The simplest and most direct method for editing xrefs is to open the source file for the referenced drawing in a separate window. This method gives you access to all objects in the referenced drawing.

Edit an xref or its nested xrefs with the External References palette or the XOPEN command. Select the xref and open its source file in a separate window. Save the edits, close the drawing, resume working and save back edited referenced drawings on page 1202 in your original drawing.

See also:

- Edit Referenced Drawings and Blocks with Nesting, OLE, or Attributes on page 1203
- Edit Selected Objects in Referenced Drawings and Blocks on page 1196

**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.

**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.

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.
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.

See also:
- Edit a Referenced Drawing in a Separate Window on page 1195
- Edit Referenced Drawings and Blocks with Nesting, OLE, or Attributes on page 1203

**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**

**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**
Controls the display of solid object shading in the current viewport.

**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 added to the working set are displayed as faded in the drawing.

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 added to the reference when changes are saved back. If you create or delete objects, they are automatically added to or removed from the working set. For example, if you use ERASE while editing a reference in place, the erased objects are removed from the working set. You can tell whether an object is in the working set or not by the way it is displayed; a faded object is not in the working set.

Use the Refedit Ribbon Contextual Tab

If you select a reference to edit in-place when the ribbon is active, the Edit Reference ribbon contextual tab is displayed. The name of the selected reference is displayed on the contextual tab. The editing buttons on the contextual tab (Add Objects to the Working Set, Remove Objects from the Working Set, Discard Changes to Reference, and Save Back Changes to Reference) are active only during in-place reference editing. The Refedit ribbon contextual tab is dismissed automatically after changes made to the reference are saved back or discarded.

Use the Refedit Toolbar

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 Objects to the Working Set, Remove Objects from the Working Set, Discard Changes to Reference, and Save Back Changes to Reference) 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.

**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 ➤ Xref And Block In-Place Editing ➤ Save Reference

**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**

### 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 1195
- Edit Selected Objects in Referenced Drawings and Blocks on page 1196

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, an error message is displayed. There are several options available to you if this occurs.

The program stores the path of the drawing that is used to create the referenced drawing. Each time you load or plot the drawing or use the xref Reload option in the External References palette to update the xref, the program checks this path to determine the name and location of the referenced drawing file. If the name or location of the drawing file has changed, the program cannot reload the xref.

If the program cannot load an xref when it is loading your drawing, it displays an error message. In this example, the program can’t find the xref HOUSE:

"\acadlt\dwgs\house.dwg": Can't open file

** Error resolving Xref HOUSE.

For each insertion of the referenced drawing, the program displays text (at the location, scale, and rotation angle of the original reference) that contains the 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 window, a task dialog box might be displayed which allows you to ignore all missing reference files or update their locations. You can use the External References paletteto update the locations of the unresolved references.

One way to avoid these errors is to be sure that when you give other people files that have xrefs attached, you also give them all the referenced files.
Change Nested Xref Paths

When the drawing is reopened and the 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
- Support search paths defined on the Files tab in the Options dialog box
- Start In folder specified in the Microsoft® Windows® application shortcut

This ensures 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 path has changed.

For example, suppose that the xref tree of the current drawing A is A>B>C, and the owner of drawing B changes the path of xref C to point to C1.dwg. When drawing A is reopened, it reflects the path change in drawing B and displays C1.dwg. However, if C1.dwg is not found, the program looks for xref C at the last location it was saved in drawing A.

See also:

- Update Referenced Drawing Attachments on page 1186

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 reference file 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, dimensioning 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 ($$) separated by a number (usually zero): for example, the referenced layer, STAIR|STEEL, becomes STAIR$$0$STEEL. You can then use the RENAME command to change STAIR$$0$STEEL 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 drawing**

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 Tools tab ➤ Drawing Utilities panel ➤ 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**

RENAM

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.

The program can maintain a log of its 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 block 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 Block from an Xref Log File

This title block 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 prints the nesting level of all affected xrefs immediately following the title block. 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
```

1210 | Chapter 31   Reference Other Drawing Files
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
Provides a high-level description of using demand loading to improve performance when working with large referenced drawings.

The program uses demand loading and saving drawings with indexes to increase performance with large referenced drawings that have been clipped using the program or that have many objects on frozen layers. With demand loading, the program loads into memory only the data from the reference drawing that is necessary to regenerate the current drawing. In other words, referenced material 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**

When a DWG reference (xref) is unloaded from the current drawing, the drawing opens much faster and uses less memory.

When an 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 pointer to the reference 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
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. At the Command prompt, enter INDEXCTL.
2. Enter 1, 2, or 3, depending on how you want to save the drawing.
   - INDEXCTL set to 1 creates a layer index. INDEXCTL set to 2 creates a spatial index. INDEXCTL set to 3 creates both layer and spatial indexes.
   - INDEXCTL set to 0 (the default) does not create an index.

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.

TREEDPTH

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.

To set the file path for xref copies

1. At the Command prompt, enter xloadpath.
2. Enter the name of the folder to use for storing temporary copies of xref files (typically a folder on your own computer).
Quick Reference

Commands
EXTERNALREFERENCES
  Opens the External References palette.

System Variables
XLOADPATH
  Creates a path for storing temporary copies of demand-loaded xref files.
Link and Embed Data (OLE)

With the Microsoft Windows OLE feature, you can copy or move information from one application to another while retaining the ability to edit the information in the original application.

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 LT drawing, or you can create an AutoCAD LT 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:
- [Use Windows Cut, Copy, and Paste](#) on page 551

**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 Blocks & References 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 LT 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 LT 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 LT 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 LT 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 576

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 LT 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.
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

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.

Import Other File Formats

You can import files, other than DWG files, that were created with other applications into your drawings.

Convert DXF Files to DWG Format

A DXF (drawing interchange format) file is a type of drawing interchange files used to transfer data between various applications.

A DXF (drawing interchange format) file is either an ASCII or a binary representation of a drawing file. It is often used to share drawing data between other CAD programs.

You can convert a DXF 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).
Find and select the DXF file you want to import, or enter the name of the DXF file at File Name.

Click Open.

Quick Reference

Commands

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.

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 line styles as much as possible
- 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.

To import a MicroStation DGN drawing

1. Click Blocks & References 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.

Quick Reference

Commands

DGNIMPORT

Imports the data from a DGN file into a new DWG file.

System Variables

DGNIMPORTMAX

Sets the maximum number of elements that are translated when importing a DGN file.

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 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 select a linetype from .lin file
1  In the Modify DGN Mapping Setup dialog box, click Linetype tab.
2  Click the drop-down list in the cell on the right-hand column.
3  Click Other... button.
4  Click Load button to select a linetype from the acad.lin (acadiso.lin) file.

If several linetypes are loaded in the Load or Reload Linetypes dialog box, select a single linetype in the Select Linetype dialog box to populate the current cell. After loading several linetypes, the additional linetypes are added to the
drop-down list so you can just select from the drop-down list for the other cells.

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

DGNMAPPING
Specifies the location of the dgnsetups.ini file where DGN mapping setups are stored.

System Variables

DGNMAPPINGPATH
Specifies the location of the dgnsetups.ini file where DGN mapping setups are stored.

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 874

To insert a WMF file

1. Click Blocks & References tab ➤ Import panel ➤ WMF File.

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
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. 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.

NOTE Although underlay files are reproductions of their source drawing, they are not as precise as drawing files. Underlays may show slight discrepancies in precision.

See also:

- For information on identifying referenced underlays, see Highlight External References in a Drawing in Attach and Detach Referenced Drawings on page 1176.

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
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, 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.

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.

<table>
<thead>
<tr>
<th>MicroStation Term</th>
<th>AutoCAD Term</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>AccuSnap</td>
<td>Osnap</td>
<td>Drafting tool for automatic snaps.</td>
</tr>
<tr>
<td>ACS</td>
<td>UCS</td>
<td>Coordinate system acronyms.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>ACS = Auxiliary Coordinate System</td>
</tr>
<tr>
<td></td>
<td></td>
<td>UCS = User Coordinate System</td>
</tr>
</tbody>
</table>

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<table>
<thead>
<tr>
<th>MicroStation Term</th>
<th>AutoCAD Term</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>ByLevel setting</td>
<td>BYLAYER setting</td>
<td>Setting that controls whether color, line weight, and line style are set for each level (layer).</td>
</tr>
<tr>
<td>Cell libraries</td>
<td>N/A</td>
<td>No reference in AutoCAD terminology.</td>
</tr>
<tr>
<td>Cells: shared and normal</td>
<td>Blocks</td>
<td>In AutoCAD, all blocks behave like shared cells. There is no reference in AutoCAD terminology to a normal cell.</td>
</tr>
<tr>
<td>Design model</td>
<td>Model space</td>
<td>DWG workmode and AutoCAD only allow for a single model. MicroStation DGN workmode allows for multiple models.</td>
</tr>
<tr>
<td>DWG file</td>
<td>DWG file</td>
<td>Native file format for each program.</td>
</tr>
<tr>
<td>Drop Element</td>
<td>Explode</td>
<td>Command used to demote element/object types to lower level. For example, cells/blocks can be demoted to geometry.</td>
</tr>
<tr>
<td>Element Attributes</td>
<td>Properties</td>
<td>Name for characteristics of elements/objects.</td>
</tr>
<tr>
<td>Fit View</td>
<td>Zoom extents</td>
<td>Command for zooming in on all elements currently in the drawing.</td>
</tr>
<tr>
<td>Handles</td>
<td>Grips</td>
<td>Vertices on geometry that can be selected and manipulated.</td>
</tr>
</tbody>
</table>

MicroStation’s “Fit View” tool fits only visible geometry.

“Zoom Extents” area includes layers that have display turned off.
## MicroStation to AutoCAD Terms

<table>
<thead>
<tr>
<th>MicroStation Term</th>
<th>AutoCAD Term</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Key-in</td>
<td>Command Line</td>
<td>Place for entering commands/variables manually.</td>
</tr>
<tr>
<td>Levels</td>
<td>Layers</td>
<td>Organizational structure used to control the appearance of objects in a drawing.</td>
</tr>
<tr>
<td>Line styles</td>
<td>Linetypes</td>
<td>Setting used to control the appearance of the line work in a drawing.</td>
</tr>
<tr>
<td>Merge into Master</td>
<td>Bind XREF</td>
<td>Inserting an external reference into the current drawing.</td>
</tr>
<tr>
<td>Message Center</td>
<td>Text Window</td>
<td>Place for viewing text messages from program feedback.</td>
</tr>
<tr>
<td>Parasolid</td>
<td>ACIS</td>
<td>Native 3D modeling kernel for each program.</td>
</tr>
<tr>
<td>Patterning</td>
<td>Hatching</td>
<td>Filling of a defined area with a defined pattern.</td>
</tr>
<tr>
<td>Pen tables</td>
<td>Plot styles</td>
<td>Used to control how linework appears when printing or plotting.</td>
</tr>
<tr>
<td>References</td>
<td>References: attachments, overlays, and underlays</td>
<td>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 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.</td>
</tr>
<tr>
<td>Seed file</td>
<td>Template drawing file</td>
<td>Files are used as a starting point for newly created files and store commonly used settings.</td>
</tr>
</tbody>
</table>
### MicroStation to AutoCAD Terms

<table>
<thead>
<tr>
<th>MicroStation Term</th>
<th>AutoCAD Term</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sheet model</td>
<td>Drawing layout (paper space)</td>
<td>Commonly used to control the output of a drawing for plotting.</td>
</tr>
<tr>
<td>Smart Line</td>
<td>Polyline</td>
<td>Multi-segmented lines.</td>
</tr>
<tr>
<td>Tags</td>
<td>Attributes</td>
<td>Element used to store textual information in a cell (block).</td>
</tr>
<tr>
<td>View</td>
<td>Viewport</td>
<td>Used to control the visibility of different sections of a drawing.</td>
</tr>
<tr>
<td>View Attributes</td>
<td>Drafting settings</td>
<td>There is no direct reference in AutoCAD terminology. Similar options can be found in the taskbar/drafting settings area of AutoCAD.</td>
</tr>
<tr>
<td>Working units</td>
<td>Drawing units</td>
<td>Used to control command input values that are dependent on unit values.</td>
</tr>
</tbody>
</table>

### 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.

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 1264.

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 LT 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.
Select the PDF underlay to adjust.

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 1190
- Clip Raster Images on page 1283

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 Edit Objects with Grips on page 556

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

Turns the display of frames on and off for all external references, images, and DWF, DWFx, PDF, and DGN underlays.

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
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
- 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 1181.

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 file where 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 LT 2011 ➤ 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 starting with AutoCAD LT 2011.

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

<table>
<thead>
<tr>
<th>Type</th>
<th>Description and versions</th>
<th>File extension</th>
</tr>
</thead>
<tbody>
<tr>
<td>BMP</td>
<td>Windows and OS/2 bitmap format</td>
<td>.bmp, .dib, .rle</td>
</tr>
<tr>
<td>CALS-I</td>
<td>Mil-R-Raster I</td>
<td>.gp4, .mil, .rst, .cg4, .cal</td>
</tr>
<tr>
<td>FLIC</td>
<td>FLIC Autodesk Animator Animation</td>
<td>.flc, .fli</td>
</tr>
<tr>
<td>GeoSPOT</td>
<td>GeoSPOT (BIL files must be accompanied with HDR and PAL files with correlation data, in the same directory)</td>
<td>.bil</td>
</tr>
<tr>
<td>IG4</td>
<td>Image Systems Group 4</td>
<td>.ig4</td>
</tr>
<tr>
<td>JFIF or JPEG</td>
<td>Joint Photographics Expert Group</td>
<td>.jpg or .jpeg</td>
</tr>
<tr>
<td>PCX</td>
<td>Picture PC Paintbrush Picture</td>
<td>.pcx</td>
</tr>
<tr>
<td>PICT</td>
<td>Picture Macintosh Picture</td>
<td>.pct</td>
</tr>
<tr>
<td>PNG</td>
<td>Portable Network Graphic</td>
<td>.png</td>
</tr>
</tbody>
</table>
Supported image file formats

<table>
<thead>
<tr>
<th>Type</th>
<th>Description and versions</th>
<th>File extension</th>
</tr>
</thead>
<tbody>
<tr>
<td>RLC</td>
<td>Run-Length Compressed</td>
<td>.rlc</td>
</tr>
<tr>
<td>TARGA</td>
<td>True Vision Raster-Based Data Format</td>
<td>.tga</td>
</tr>
<tr>
<td>TIFF</td>
<td>Tagged Image File Format</td>
<td>.tif or .tiff</td>
</tr>
</tbody>
</table>

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 95.

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.

See also:

- For information on identifying referenced images, see Highlight External References in a Drawing in Attach and Detach Referenced Drawings on page 1176

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 1173.

To attach an image

1. Click Blocks & References tab ➤ References panel ➤ Image.
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 Blocks & References tab ➤ References panel ➤ Image.
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 Blocks & References tab ➤ References panel ➤ Image.
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.

**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 1190
- Clip Underlays on page 1264

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 Edit Objects with Grips on page 556.

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.

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**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 Blocks & References tab ➤ References panel ➤ Image.
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 LT 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 1181.

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

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.

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.

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 LT 2008 (and later releases) in AutoCAD LT 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 objects maintain visual fidelity when they are viewed in AutoCAD LT 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 1406. When visual fidelity is on, annotative objects are decomposed and scale representations are saved (in an anonymous block on page 1388) to separate layers. These layers are named based on their original layer and appended with a number. If you explode the block in AutoCAD LT 2007 (or earlier releases), and then open the drawing in AutoCAD LT 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 LT 2008 (and later releases) in AutoCAD LT 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 LT 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 LT 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 LT 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 LT 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 LT 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.

**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.

**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 LT 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 181.

**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 LT 2011 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.

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 LT 95
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 create a sheet set and insert LispVariables and SheetSet
Manager fields. The LispVariables and SheetSet Manager fields are not available
in AutoCAD LT. The drawings created in AutoCAD that contain LispVariables
or SheetSet Manager fields 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 LT 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.

**WARNING** Once you use the SHADEMODE command in AutoCAD LT to turn solid object shading off for an object created in AutoCAD, you cannot turn it back on except by using the UNDO command. If you make changes to the object, you can turn the shading on again only in AutoCAD.
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.

Turn Off Perspective View in an AutoCAD Drawing

Set the PERSPECTIVE system variable to 0 to turn off perspective view in an AutoCAD drawing that is open in AutoCAD LT. You cannot turn on perspective view in a drawing that is open in AutoCAD LT.

Quick Reference

System Variables

FIELDDISPLAY

Controls whether fields are displayed with a gray background.
PERSPECTIVE

Specifies whether the current viewport displays a perspective view.

POINTCLOUDDENSITY

Controls the number of points displayed at once for all point clouds in the drawing view.

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 LT 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 LT 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.
Access External Databases

You can associate, or link, data stored in external databases with graphical objects in a drawing.

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).

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 on page 7
Collaborate with Others
You can collaborate on design projects using passwords and digital signatures. When a drawing has a password, it cannot be viewed until the password is entered. When a drawing is signed, the signature identifies an individual or an organization through a digital ID (certificate).

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.

In AutoCAD LT®, you can maintain existing passwords. AutoCAD LT is required for adding or removing passwords.
The primary use of encryption is to help prevent drawing data from being stolen. Encryption can also be used to help keep data confidential.

**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.

### 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.

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.
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.

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
- 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 LT 2011 ➤ 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.

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Specifies password or digital signature options for a drawing file.
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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 LT 2011. 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 ADSKSIPTIMESRVR 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.

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   Specifies password or digital signature options for a drawing file.
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signature is opened.

Utilities
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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.
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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.

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Attach Digital Signatures
Attaches a digital signature to files.
Use the Internet for Collaboration

You can access and store drawings and related files on the Internet.

Get Started with Internet Access

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).

In this text, the term Internet is used to refer to both the Internet and an intranet. 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.

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 LT 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 LT 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 Blocks & References 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 1336).

2. Click Blocks & References 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 Blocks & References 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 Blocks & References 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 Tools menu ➤ Options.
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 Tools tab ➤ Drawing Utilities panel ➤ Drawing Properties.
2 On the Summary tab, enter a relative path in Hyperlink Base.
3 Click OK.

To create a relative hyperlink

1 Click Blocks & References 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.

**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.

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**Quick Reference**

**Commands**

ATTACHURL

Attaches hyperlinks to objects or areas in a drawing.

DETACHURL

Removes hyperlinks in a drawing.
GOTOURL
Open 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

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, and create a transmittal package of drawings that automatically includes all related files.

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 LT to open and save files from the Internet. The AutoCAD LT file input and output commands (OPEN, EXPORT, and so on) recognize any valid URL path to an AutoCAD LT file. The drawing file that you specify is downloaded to your computer and opened in the AutoCAD LT 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 an 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 an 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 an 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 an 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 an 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.
3. 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 as described in To open an AutoCAD LT file from the Internet by browsing an FTP site.
4. Double-click one of the FTP sites, and select a file.
5. 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 LT 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 LT 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
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
- 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

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 plot and text window 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.

With eTransmit, you can send a transmittal set of DWG files to a Buzzsaw site.

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 LT 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 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.

Quick Reference

Commands
OPEN
Opens an existing drawing file.

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.

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 includes two tabs. Using these tabs, you can view and change the files to be 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.

<table>
<thead>
<tr>
<th>File Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>*.dwg</td>
<td>Root drawing file and any attached external references</td>
</tr>
<tr>
<td>Various image file types</td>
<td>Raster image files that are attached externally to the root drawing or external referenced drawings, or as part of a material</td>
</tr>
<tr>
<td>*.dgn</td>
<td>DGN files that are attached externally to the root drawing or external referenced drawings</td>
</tr>
<tr>
<td>*.dwf</td>
<td>Design Web Format files that are attached externally to the root drawing or external referenced drawings</td>
</tr>
<tr>
<td>*.dwfx</td>
<td>DWF file with XPS information that is attached externally to the root drawing or an externally referenced drawing</td>
</tr>
<tr>
<td>*.pdf</td>
<td>PDF files that are attached externally to the root drawing or external referenced drawings</td>
</tr>
<tr>
<td>*.fmp</td>
<td>Font Mapping File used for the In-Place Text Editor when working with Multiline Text objects</td>
</tr>
<tr>
<td>*.ctb</td>
<td>Color-dependent plot style files used to control the appearance of the objects in the drawings of the transmittal set when plotting</td>
</tr>
<tr>
<td>*.stb</td>
<td>Named plot style files used to control the appearance of the objects in the drawings of the transmittal set when plotting</td>
</tr>
</tbody>
</table>
### File Types Automatically Added to a Transmittal Package

<table>
<thead>
<tr>
<th>File Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>*.pc3</td>
<td>Plotter configuration files used to control the device and settings for the device when plotting the layouts of the drawings in the transmittal set</td>
</tr>
</tbody>
</table>

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

<table>
<thead>
<tr>
<th>File Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>*.shx</td>
<td>Compiled shape files used for text styles or complex linetypes</td>
</tr>
<tr>
<td>*.ttf</td>
<td>True Type font files used for text styles</td>
</tr>
<tr>
<td>*.pfa, *.pfb</td>
<td>PostScript Type 1 font files</td>
</tr>
<tr>
<td>*.pat</td>
<td>Hatch pattern files</td>
</tr>
<tr>
<td>*.lin</td>
<td>Linetype definition files</td>
</tr>
<tr>
<td>*.pmp</td>
<td>Plotter model parameter files, used to store configuration information specific to a plotter such as custom paper sizes</td>
</tr>
<tr>
<td>*.unt</td>
<td>Unit conversion file, used by QuickCalc</td>
</tr>
</tbody>
</table>

**OLE objects (linked files)**

OLE objects that are linked to files in the drawing 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.

**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 LT 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 LT 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 hexadecimal values below 80 before using eTransmit.

**To create a transmittal package in a folder you specify**

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 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 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

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](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.

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**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](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*.

Use the Publish to Web Wizard to Create Web Pages | 1365
See also:
  - DesignCenter on page 88

To use the Publish to Web wizard

1. Click Output tab ➤ Publish panel ➤ 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/.

You can access Autodesk Seek in the following ways:
  - From the Autodesk Seek home page http://seek.autodesk.com
On the ribbon Insert tab (to search) or the Output tab (to share)

Click Application menu ➤ Publish ➤ Share with Autodesk Seek

Click Share with Autodesk Seek from a selected block shortcut menu

From the menu bar, click File ➤ Share with Autodesk Seek

From the menu bar, click Tools menu ➤ Toolbars ➤ Autodesk Seek

From the Autodesk Seek toolbar, click Autodesk Seek

Click Autodesk Seek Design Content from DesignCenter (ADCENTER)

At the Command prompt, enter SHAREWITHSEEK to share and SEEK to search

With Autodesk Seek, you can upload AutoCAD drawing files and blocks under the categories Architecture, Civil Engineering, and Structural Engineering. This facilitates searching that results in significant time savings during production.

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 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.
   Follow the prompts on the web page and enter all required, and any optional information, to complete the Share with Autodesk Seek process.

To search for blocks and drawings in Autodesk Seek

1 Click Insert tab ➤ Content panel ➤ Seek Design Content.

2 Type one or more terms in the text box and click the Search button.
   The Autodesk Seek website is displayed with the results for the terms you searched.

Quick Reference

Commands

SEEK
   Opens a web browser and displays the Autodesk Seek home page.
SHAREWITHSEEK
   Uploads blocks or drawings to the Autodesk Seek website.
Use Markups for Design Review

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 LT 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 1363.

When you open the marked-up DWF or DWFx file in AutoCAD LT®, you can view the redline markups in the AutoCAD LT 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 1363
**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 1363
Quick Reference

Commands

PUBLISH

Publishes drawings to electronic sheet sets (DWF, DWFx, or PDF files) or plotters.

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 1363

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 LT. 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 LT 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 LT 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 ensures 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 LT 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 LT. Marked-up DWF or DWFx files originally created from earlier versions of AutoCAD LT can be opened in Autodesk Design Review.

See also:
- Review and Markup Files with Design Review on page 1363

**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 LT 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.

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.
In the Markup Set Manager, click a markup node to view the redline geometry in the drawing area.

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 1363

**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 1363

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.
Render Drawings
Draw 2D Isometric Views

The Isometric Snap/Grid mode helps you create 2D isometric 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.
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.

ISPLANE
Specifies the current isometric plane.

ORTHO
Constrains cursor movement to the horizontal or vertical direction.

SNAP
Restricts cursor movement to specified intervals.

System Variables

ISOLINES
Specifies the number of contour lines per surface on objects.

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.
Glossary

Commands associated with definitions are shown in parentheses at the end of the definition.

**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 bar** Toolbar-like UI that displays the actions associated with a parameter object.

**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.

**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 acadlt.pgp file.

**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)

**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.

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.

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 LT library search path The order in which AutoCAD LT looks for a support file: current directory, drawing directory, directory specified in the support path, and directory containing the AutoCAD LT executable file, acadlt.exe.

AutoCAD LT 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)

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.

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.

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 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.

**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 (*acadlt.cui*).

**BYBLOCK** A special object property used to specify that the object inherits the color or linetype of any block containing it. *See also* BYLAYER.

**BYLAYER** A special object property used to specify that the object inherits the color or linetype associated with its layer. *See also* BYBLOCK.

**candela** The SI unit of luminous intensity (perceived power emitted by a light source in a particular direction) (Symbol: cd). Cd/Sr

**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.

circular external reference  An externally referenced drawing (xref) that references itself directly or indirectly. The xref that creates the circular condition is ignored.

CMYK  For cyan, magenta, yellow, and key color. A system of defining colors by specifying the percentages of cyan, magenta, yellow, and the key color, which is typically black.

Color bleed scale  Increases or decreases the saturation of the reflected color from the material.

color map  A table defining the intensity of red, green, and blue (RGB) for each displayed color.

column  A vertically adjacent table cell selection spanning the height of the table. A single column is one cell in width.

command line  A text area reserved for keyboard input, prompts, and messages.

compass  
A visual aid that indicates the directions North, South, East, and West in the current model.

constraint bar  Displays the geometric constraints associated with objects or with points on objects.

constraint point  Point on an object that can be geometrically and/or dimensionally constrained (for example, an endpoint or an insertion point).

constraints  Form of parametric design.
Rules that govern the position, slope, tangency, dimensions, and relationships among objects in a geometry.

construction plane  See workplane.

contextual ribbon tab  
A ribbon tab that is displayed only when a particular type of object 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)
**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.

**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.

**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** 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 (CUI) file An XML-based file that stores customization data. You modify a customization file through the Customize User Interface dialog box. CUI files replace MNU, MNS, and MNC files that were used to define menus in earlier releases.

data link A connection between a table and an external source of data.

decimal degrees A notation for specifying latitude and longitude. For example, 35.1234°, 100.5678°.
Latitude always precedes longitude.

default drawing See initial environment.

default 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.

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.

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.

**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 LT window and on how many toolbars and other elements are displayed. See also AutoCAD LT 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](image)

**drawing limits**  See grid limits.

**drawing set**  A collection of drawings assembled using the Publish dialog box.

**drawing template**  A drawing file with preestablished settings for new drawings such as acadlt.dwt and acadltiso.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.

**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.

**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)
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.

enterprise customization file  A CUI file that is typically controlled by a CAD manager. It is often accessed by many users and is stored in a shared network location. The file is read-only to users to prevent the data in the file from being changed. A CAD manager creates an enterprise CUI file by modifying a main CUI file and then saving the file to the support location defined in the Options dialog box, Files tab.

environment map  A bitmap that is used to simulate reflections in materials that have reflective properties. The map is “wrapped” around the scene and any reflective object will show the appropriate portion of the map in the reflective parts of its material.

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)

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.

fill  A solid color covering an area bounded by lines or curves. (FILL)

filters  See coordinate filters.
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.

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)

general property Properties that are common between a selection of objects. These include Color, Layer, Linetype, Linetype scale, Plot style, Lineweight, Hyperlink, and Thickness.

geographic elevation The relative height along the specified up-direction defined for a geographic marker.

geographic marker Visual representation of geographic location information.

geometric constraint Rules that define the geometric relationships of objects (or points of objects) elements and control how an object can change shape or size.

Geometric constraints are coincident, collinear, concentric, equal, fix, horizontal, parallel, perpendicular, tangent, and vertical.

geometry All graphical objects such as lines, circles, arcs, polylines, and dimensions. Nongraphical objects, such as linetypes, lineweights, text styles, and layers are not considered geometry. See also named object.

graphics area See drawing area.

graphics window See AutoCAD LT window and drawing area.

grid An area covered with regularly spaced dots or lines to aid drawing. The grid spacing is adjustable. The grid dots are never plotted. See also grid limits. (GRID)
grid limits The user-defined rectangular boundary of the drawing area covered by dots when the grid is turned on. Also called drawing limits. (LIMITS)

grid limits

grip modes The editing capabilities activated when grips are displayed on an object: stretching, moving, rotating, scaling, and mirroring.

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.

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.

horizontal landing An optional line segment connecting the tail of a leader line with the leader content.

horizontal ribbon

The ribbon, when it is oriented across the top of the file window.

i-drop A method by which a drawing file can be dragged from a Web page and inserted into another drawing.

IGES For Initial Graphics Exchange Specification. A standard format for digital representation and exchange of information between CAD/CAM systems. In AutoCAD-based products, the commands to import and export IGES files are available only in AutoCAD Mechanical.

Illuminance In photometry, illuminance is the total luminous flux incident on a surface per unit area.

indirect bump scale Scales the effect of the base material’s bump mapping in areas lit by indirect light.

initial environment The variables and settings for new drawings as defined by the default drawing template, such as acadlt.dwg or acadltiso.dwg. See also template drawing.
interface element A user interface object that can be customized, such as a toolbar, pull-down menu, shortcut key, dockable window, and so on.

interpolation points Defining points that a B-spline passes through. See also approximation points and fit points.

island An enclosed area within another enclosed area. Islands may be detected as part of the process of creating hatches, polylines, and regions. (BHATCH, BOUNDARY)

ISO For International Standards Organization. The organization that sets international standards in all fields except electrical and electronics. Headquarters are in Geneva, Switzerland.

isometric snap style A drafting option that aligns the cursor with two of three isometric axes and displays grid, making 2D isometric drawings easier to create.

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)

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.

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.

**lumen** The SI unit of luminous flux (Symbol: lm). Cd * Sr

**lux** The SI unit of illuminance (symbol: lx). Lm/m^2

**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 `acadlt.cui` file (the default main CUI file) is automatically loaded when you start AutoCAD LT.

**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.

**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)

**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 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.

nonassociative dimension A dimension that does not automatically change as the associated geometry is modified. Controlled by the DIMASSOC system variable. See also associative dimension and exploded dimension.

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.

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 LT Runtime Extension) A compiled-language programming environment for developing AutoCAD LT 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.

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)

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.

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.

personalization Customizes the executable file during installation with the user name, company, and other information.
**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.

**plan view** A view orientation from a point on the positive $Z$ axis toward the origin (0,0,0). (PLAN)

**planar projection** Mapping of objects or images onto a plane.

**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.

**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 1405.
polar tracking A precision drawing tool that displays temporary alignment paths defined by user-specified polar angles. See also Polar Snap.

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)

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.

prompt A message on the command line or in a tooltip that asks for information or requests action such as specifying a point.

proxy object A substitute for a custom object when the ObjectARX application that created the custom object is not available. See also custom object and object enabler.

push pin A push pin-shaped button used on the ribbon and in the application menu. On the ribbon, push pins are used to keep a ribbon panel expanded. In the application menu, push pins keep an item in the list of recently opened items.

PWT A template file format used to publish drawings to the Web.

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)

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 are 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.

rewind  Restores the previous view or movement path created by SteeringWheels and other navigation tools.

RGB  For red, green, and blue. A system of defining colors by specifying percentages of red, green, and blue.

ribbon  A palette that displays buttons and controls used for both 2D drawing and annotation and 3D modeling, viewing, and rendering. See also ribbon tab and ribbon panel and slide-out panel. (RIBBON)

ribbon panel  A 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.

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)

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 set One or more selected objects that a command can act upon at the same time.

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.

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.

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.

STB file  For plot style table file. Contains plot styles and their characteristics.

SteeringWheels  Tool set that provides access to 2D navigation tools.

surface associativity  See associative surfaces

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 a 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.
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.

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 icon An icon that indicates the orientation of the UCS axes. (UCSICON)
underconstrained geometry  Objects with unsolved degrees of freedom are underconstrained.

underlay A DWF, or DGN file used to provide visual context in a drawing file. Underlays cannot be edited, and do not provide the full range of notification. Underlays cannot be bound to a drawing. See also external reference (xref).

up direction A vector defining what direction is Up. By default this is the positive Z – axis (0,0,+1).

The up direction and the north direction are always constrained such that they are perpendicular to each other.

user coordinate system (UCS) A user-defined coordinate system that defines the orientation of the X, Y, and Z axes in 3D space. The UCS determines the default placement of geometry in a drawing. See also world coordinate system (WCS).

user parameter Named user-defined variable (real number or an expression) that can be used in expressions for dimensional constraints or other user parameters.

UVW The material’s coordinate space. Used instead of XYZ because that is usually reserved for the world coordinate system (WCS). Most material maps are a 2D plane assigned to a 3D surface. The U, V, and W coordinates parallel the relative directions of X, Y, and Z coordinates. If you look at a 2D map image, U is the equivalent of X, and represents the horizontal direction of the map. V is the equivalent of Y, and represents the vertical direction of the map. W is the equivalent of Z and represents a direction perpendicular to the UV plane of the map.

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. (VPOINT, VIEW)

viewpoint The location in 3D model space from which you are viewing a model. See also view and viewport. (VPOINT)

viewport A bounded area that displays some portion of the model space of a drawing. The TILEMODE system variable determines the type of viewport created. 1. When TILEMODE is off (0), viewports are objects that can be moved and resized on a layout. (MVIEW) 2. When TILEMODE is on (1), the entire drawing area is divided into nonoverlapping model viewports. See also TILEMODE, view, and viewpoint. (VPORTS)

viewport configuration A named collection of model viewports that can be saved and restored. (VPORTS)

virtual screen display The area in which the program can pan and zoom without regenerating the drawing.

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.

working drawing A drawing for manufacturing or building purposes.

workplane Another name for the XY plane of the user coordinate system. See also elevation and user coordinate system (UCS).
**workspace** A set of menus, toolbars and dockable windows (such as the Properties palette, DesignCenter, and the Tool palettes window) that are grouped and organized so that you can work in a custom, task-oriented drawing environment.

**world coordinate system (WCS)** A coordinate system used as the basis for defining all objects and other coordinate systems. See also user coordinate system (UCS).

**world coordinates** Coordinates expressed in relation to the world coordinate system (WCS).

**wrap around**

Behavior where the cursor wraps around the window and appears on the opposite side to allow the continuation of a drag operation instead of stopping at the edge of the drawing area.

**X,Y,Z point filters** See coordinate filters.

**xref** See external reference (xref).

**zoom** To reduce or increase the apparent magnification of the drawing area. (ZOOM)
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