AutoCAD®

Autodesk®

# User's Guide

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# Find the Information You Need

The Help system is organized in a structured design that makes information easy to locate.

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

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.

## In this chapter


- Install the Product
- Use the Help System Efficiently
- Use Quick Help on the Info Palette
- Learn the Product
- Access Subscription Center
- Receive Product Updates and Announcements
- View the Product Readme

# Install the Product

You can easily install and configure this program on a stand-alone computer.

Insert the product CD in the CD-ROM drive. Then, in the Media Browser, on the Install tab, click Stand-Alone Installation. Then, click Install. The Installation wizard leads you through the installation process. The Media Browser also provides instructions for network installations and access to information about technical support and licensing.

If you want more information about installing a stand-alone version of AutoCAD, consult the *Stand-Alone Installation Guide*. To access this guide, in the Media Browser, click the Documentation tab. Then click *Stand-Alone Installation Guide* (*.pdf*).

If you want more information about deploying this program on a network, consult the *Network Administrator's Guide*. To access this guide, in the Media Browser, click the Documentation tab. Then, click Network Administrator's Guide (*.pdf*).

Once the product is installed, you can access either the *Stand-Alone Installation Guide* or the *Network Administrator's Guide* in the Help system.

### Migration Tools

Migration Tools help you when upgrading to a new release of this program. You can download the Migration Tools in several languages from the Autodesk website.

---

**NOTE** You can migrate some custom settings and files by using the Migrate Custom Settings dialog box. For more information about the Migrate Custom Settings dialog box, see the *Stand-Alone Installation Guide*.

---

Available tools include

- Layer State Converter
- Batch Drawing Converter
- AutoLISP Compatibility Analyzer
- ScriptPro

Go to the Autodesk website,
*http://www.autodesk.com*
, and perform a search on Migration Tools.

**NOTE** Because these are public tools, product support for them is limited to the Autodesk discussion groups.

# Use the Help System Efficiently

You can get much more benefit from the Help system when you learn how to use it efficiently.

The Help system contains complete information about using this program. In the Help window, you use the left pane to locate information. The tabs above the left pane give you several ways for finding the topics you want to view. The right pane displays the topics you select.

## Find Information in Help

The tabs on the left side of the Help window provide different methods for finding information.

To locate a specific word or phrase in the current topic, click in the topic text and use the CTRL+F keys.

### Contents Tab

- Presents an overview of the available documentation in a list of topics and subtopics.
- Allows you to browse by selecting and expanding topics.
- Provides a structure so you can always see where you are in Help and quickly jump to other topics.

### Index Tab

- Displays an alphabetical list of keywords related to the topics listed on the Contents tab.
- Accesses information quickly when you already know the name of a feature, command, or operation, or when you know what action you want the program to perform.

### Search Tab

- Provides full-text search of all the topics listed on the Contents tab.
- Allows you to perform an exhaustive search for a specific word or phrase.

- Displays a ranked list of topics that contain the word or words entered in the keyword field.

### Ask Me Tab

- Allows you to find information using a question phrased in everyday language.
- Displays a ranked list of topics that correspond to the word or phrase entered in the question field.
- Provides a "Search the Web" link that sends a query to a search engine on the Web.

### To start Help

- Click Help menu ➤ Help, or press F1.

  For Developer Help, click Help ➤ Additional Resources ➤ Developer Help.

---

**NOTE** You can press F1 at the Command prompt, in a dialog box, or at a prompt within a command to display Help information.

---

### To use the Help Contents

1. If necessary, click the Show button to display the left pane of the Help window. Then click the Contents tab to display Help Contents.
2. To expand the Help Contents list, use one of these methods:
   - Double-click a closed-book icon or click the + icon next to it.
   - Right-click in Help Contents. Click Open All.
3. To close the Help Contents list, use one of these methods:
   - Double-click an open-book icon or click the - icon next to it.
   - Right-click in Help Contents. Click Close All.
4. To view a topic, use one of these methods:
   - In Help Contents, click the topic.
   - In a topic, click any blue underlined text.

# Use Searches

A basic search using the Search tab consists of the word or phrase you want to find.

The basic search rules are as follows:

- Type your search in uppercase or lowercase characters; searches are not case-sensitive.
- Search for any combination of letters (a-z) and numbers (0-9).
- Do not use punctuation marks such as the period, colon, semicolon, comma, hyphen, and single quotation marks; they are ignored during a search.
- Group the elements of your search using double quotation marks or parentheses to set each element apart.

---

**NOTE** When you use the Search tab, all topics that contain the word or phrase that you specify are listed. This includes the text in topic titles and labels, not just the topic text.

---

### Use Advanced Search

Advanced full-text search on the Search tab allows you to search using Boolean operators and wild cards. You can limit the search to previous results, match similar words, or search topic titles only. When searching for multi-word topics, use double quotation marks (" ") to enclose words that must appear next to each other in the specified sequence. For example, enter "**specifying units of measurement**" to find only topics with all those words in that order. If you don't use the quotation marks around that text, Help finds all topics containing any one of the listed words, that is, all topics containing "specifying", all topics containing "units", all topics containing "of", and all topics containing "measurement".

With the AND, OR, NOT, and NEAR operators, you can precisely define your search by creating a relationship between search terms. The following table shows how you can use each of these operators. If no operator is specified, AND is used. For example, the query "spacing border printing" is equivalent to "spacing AND border AND printing".

| Search for | Example | Results |
|---|---|---|
| Both terms in the same topic | "tree view" AND "palette" | Topics containing both the words "tree view" and "palette" |

| Search for | Example | Results |
|---|---|---|
| Either term in a topic | "raster" OR "vector" | Topics containing either the word "raster" or the word "vector" or both |
| The first term without the second term | "ole" NOT "dde" | Topics containing the word "OLE," but not the word "DDE" |
| Both terms in the same topic, close together | "user" NEAR "kernel" | Topics containing the word "user" within eight words of the word "kernel" |

**NOTE** The |, &, and ! characters do not work as Boolean operators. You must use AND, OR, and NOT.

**To search for information in Help**

1. Click the Search tab. Enter the word or phrase you want to find.
2. (Optional) Refine your search with Boolean operators:
   - Click

     to add Boolean operators to your search.
   - Enter the word or phrase you want to appear after the Boolean operator.

   (Optional) Further refine your search with the check boxes at the bottom of the Search tab:
   - **Search previous results.** Refines a search by applying new search criteria to the results of a previous search.
   - **Match similar words.** Expands the search to include words similar to your search criteria, rather than limiting the search to exact matches.
   - **Search titles only.** Limits the search to topic titles, rather than searching titles and topic contents.
4. Click List Topics and select the topic you want. Click Display.
5. To sort the topic list, click the Title, Location, or Rank column heading.

To view only procedures, click the Title column heading and scroll down to the items that begin with the word “To.”

# Use Ask Me

You can find information by typing a question phrased in everyday language.

You can perform this natural language query in Help by clicking the Ask Me tab and entering a question, a word, or a phrase. For example, you could enter: **How do I create a layout?**

The results of natural language queries are often more accurate than the results of searches. Matching topics are ranked by percentages that reflect how likely they are to answer your questions.

You can also expand your natural language query to the Web.

### To use natural language queries in Help

1. On the Ask Me tab, enter the question you want answered. Alternatively, you can enter a phrase or a single word.
2. To narrow query results, select a particular item or set from the List of Components to Search.
3. Click the link for the topic you want to display.

---

**NOTE** For hints on obtaining good query results, click the Query Tips link on the Ask Me tab.

---

### To expand your queries to the Web

1. If the results of a natural language query do not provide the information you need, scroll to the bottom of the list of matching topics that resulted from your query.
2. Click the Search the Web link at the bottom of the list

A web search engine displays the results of your search.

---

**NOTE** For hints on obtaining good query results, click the Query Tips link on the Ask Me tab.

---

# Understand How Help Topics Are Organized

Most topics in this Help system have three tabs above the right pane of the Help window. The tabs display different types of information.

The tabs display different types of information.

- **Concepts tab.** Describe a feature or function. When you click a Concepts tab, the Help Contents list in the left pane of the Help window expands and highlights the current topic. The Contents tab displays the structure of the Help on that topic. You can easily display nearby topics by clicking them in the list.
- **Procedures tab.** Provides step-by-step instructions for common procedures related to the current topic. After displaying a procedure, you can click the Procedures tab to redisplay the current list of procedures.
- **Commands tab.** Lists commands and system variables related to the current topic, with links to detailed descriptions of the commands. If you click an entry on the Commands tab, the *Command Reference* opens to the command or system variable that you selected.

When you click a different tab, the topic remains the same. Only the type of information displayed—concepts, procedures, or commands links—is different.

#### Concepts Tab Organization

In the Concepts tab, there are two types of information displayed: *navigation text* and *destination text*. Navigation text displays links with short descriptions. The purpose of navigation text is to guide you step-by-step to the information that you need. The links on navigation pages lead to additional navigation pages deeper in the Help structure until you come to a destination page. Each link is designed to provide you with more detailed information.

#### Procedures Tab and Commands Tab Organization

As you navigate deeper into the Help structure on the Contents tab, the corresponding information on the Procedures tab and on the Commands tab becomes more specific, and the number of entries displayed on each of these two tabs decreases.

#### To change the type of Help information displayed

- Click the Concepts, Procedures, or Commands tab to change the type of information that is displayed in the right pane of the Help window.

## Use the Up Arrow to Navigate Help Topics

An up arrow may be displayed in the upper-right corner of a topic.

Click this arrow to move up a level from the current topic to one that contains a broader scope of information.

# Print Help Topics

The quickest way to print the current topic is to right-click within the topic and click Print.

The Print button on the Help toolbar provides these print options:

- Print the selected topic (recommended)
- Print the selected heading and all subtopics

---

**NOTE** When you select the second option, you may get numerous printed pages, depending on how many subtopics the currently selected topic contains.

---

### To print a Help topic

1. Display the topic you want to print.
2. Right-click in the topic pane. Click Print.
3. In the Print dialog box, click Print.

### To print a selected heading and all subtopics

1. Display the topic you want to print and make sure that the Contents tab is showing.
2. On the Help toolbar, click the Print button.
3. In the Print Topics dialog box, click Print the Selected Heading and All Subtopics.
4. Click OK.

# Show and Hide the Contents Pane

Use the Hide button on the Help toolbar to shrink the Help window to a compact size by hiding the pane that contains the Contents, Index, Search, and Ask Me tabs.

The compact window size is best for displaying procedures while you work.

Use the Show button to expand the Help window to display the pane that contains Help Contents, Index, Search, and Ask Me tabs. The expanded window size is best for locating and displaying conceptual and reference information.

## Get Additional Help

You can access several additional sources of help.

- **Press F1 from within a command, system variable, or dialog box.** Displays complete information from the *Command Reference*.
- **Click the question mark button in many dialog boxes.** Displays a description of the dialog box option you select.
- **View the product Readme topic in Help.** Displays late-breaking information about this product.
- **Display Quick Help on the Info Palette.** Displays procedures relevant to the current command in a compact palette.

Other resources help you get information about Autodesk products and assistance with your questions about this program.

- **Autodesk website.** Access *http://www.autodesk.com* .
- **Local support.** Check with your dealer or Autodesk country/region office.

# Use Quick Help on the Info Palette

Quick Help on the Info palette provides convenient information from the Help system. With Quick Help, you can display procedures in a compact palette that takes up very little space in your drawing area.

During any command, Quick Help displays a list of procedures that are relevant to the current command. With Quick Help, you can click a procedure to display it in the Info palette, a compact palette that takes up very little space in your drawing area.

Often, the guidance you get from Quick Help is just enough to get you started performing unfamiliar or rarely used tasks.

Normally, the Info palette updates the Quick Help information that is displayed as you start new commands. However, if you need to freeze the information displayed, you can lock the Info palette.

**To display Quick Help**

■ Click Help menu ➤ Info Palette.

**To navigate Quick Help**

1. Right-click in the Info palette.
2. Click Home, Back, or Forward to navigate topics, as you would in a web browser.

**To print Quick Help information**

1. Display the Quick Help information you want to print.
2. Right-click in the Info palette. Click Print.
3. In the Print dialog box, click OK.

**To lock Quick Help on the current topic**

1. Display the Quick Help information you want to see.
2. Right-click in the Info palette. Click Lock.

   The Quick Help information that is displayed in the Info palette is locked in place until you unlock it.

# Learn the Product

Training programs and products from Autodesk Professional Services help you learn the key technical features of your Autodesk software.

Autodesk Professional Services training programs and products help you learn the key technical features of your Autodesk software.

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

### Autodesk Authorized Training Centers

The Autodesk® Authorized Training Center (ATC® ) network delivers Autodesk-authorized, instructor-led training to design professionals who use Autodesk software. Autodesk Authorized Training Centers use experienced and knowledgeable instructors. More than 1,100 ATC sites are available worldwide to meet your needs for discipline-specific, locally based training.

To find a training center near you, contact your local Autodesk country office or visit
*http://www.autodesk.com/atc*
.

### Autodesk Official Training Courseware

Autodesk Official Training Courseware (AOTC) supports training organizations and customers. AOTC is authorized technical training material developed by Autodesk for traditional 1- to 5-day, instructor-led classroom training. AOTC courseware covers key concepts in step-by-step exercises. You can purchase AOTC from your local reseller or distributor, or you can order it online from the Autodesk Store at
http://www.autodesk.com/aotc.

### Partner Products and Services

Autodesk works together with thousands of software partners around the world. These partners provide products and services that enhance Autodesk products for design professionals. Visit the Partner Products & Services page at
*http://www.autodesk.com/partnerproducts*
for a list of resources available for your Autodesk product and your industry.

### e-Learning

Autodesk e-Learning features interactive lessons organized into product catalogs. Each lesson is 15-30 minutes in length 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 what lessons will be most helpful, and gauges learning progress.

If you are a member of Autodesk subscription, you can access e-Learning and other subscription services from within your Autodesk product. For more information about how to access e-Learning in the product, see “Access Subscription Center” on page 13. For more information about Autodesk subscription resources, visit
*http://www.autodesk.com/subscription*
.

# Access Subscription Center

The Subscription Center is available to subscription members from within the product. If you are a subscription member, you can access subscription services by using Help and clicking a Subscription resource. To learn more about Autodesk subscription membership, visit http://www.autodesk.com/subscription.

# Overview of Subscription Center

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.

By clicking the Help menu, members have access to the following options:

- **Subscription e-Learning Catalog.** Features interactive lessons organized into product catalogs. 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 what lessons will be most helpful, and gauges learning progress.
- **Create Support Request.** Provides direct one-to-one communication with Autodesk support technicians. You receive fast, complete answers to your installation, configuration, and troubleshooting questions.
- **View Support Requests.** Allows you to track and manage your questions and responses through Autodesk's state-of-the-art support system.
- **Edit Subscription Center Profile.** Allows you to set up and maintain your subscription account.

### Subscription Resources and Privacy

Subscription resources provide interactive product features over the Internet. Each time you access subscription resource (such as e-Learning or Create Support Request) from the Help menu 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 Help menu. For more information about how to install the utility, see "To install the Subscription Center (part of the CAD Manager Control utility)." For more information about using the utility once it is installed, click Help in the CAD Manager Control utility window.

#### To access the Subscription Center in the program

1. Click Help menu, and then click the subscription resource you want to access.

---

**NOTE** Subscription Center is not available to all product users. If Subscription Resources is not available on the Help menu in your product, your product is not entitled to subscription benefits.

---

#### To install the Subscription Center (part of the CAD Manager Control utility)

1. Double-click *setup.exe* on the product CD.
2. In the Media Browser, Install tab, click either Multi-Seat Stand-Alone Installation or Network Deployment.
3. Under Install Supplemental Tools, click Autodesk CAD Manager Tools 3.0.
4. Under Autodesk CAD Manager Tools 3.0, click Install.

   After you install the CAD Manager Control utility, you access it from the Start menu (Windows).

# Receive Product Updates and Announcements

The Communication Center provides up-to-date product information, software updates, product support announcements, and other product-related announcements on your screen. The types of information and the frequency of announcements can be easily configured to fit your needs.

# Overview of Communication Center

Communication Center provides the following kinds of announcements:

- **General Product Information.** Stay informed about Autodesk company news and product announcements; give your feedback directly to Autodesk.
- **Product Support Information.** Get breaking news from the Product Support team at Autodesk.
- **Subscription Information and Extension Announcements.**Receive announcements and subscription program news if you are an Autodesk subscription member (available in countries/regions where Autodesk subscriptions are offered).
- **Articles and Tips.** Be notified when new articles and tips are available on Autodesk websites.

You use the Welcome wizard to set Communication Center for your country/region, and for the frequency you prefer for updates and the information channels you want displayed.

To open Communication Center, click the Communication Center icon in the tray on the right side of the status bar.

### Communication Center Online Privacy

Communication Center is an interactive feature that must be connected to the Internet to deliver content and information. Each time Communication Center 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 Communication Center
- **Product Release Number.** The version of the product
- **Product Language.** The language version of your product
- **Country/Region.** The country/region that was specified in the Communication Center settings
- **Your Subscription Contract Number.** The information sent to Autodesk if you entered it in the Error Report dialog box

Autodesk compiles statistics using the information sent from Communication Center 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 the company's published privacy policy, which is available on http://www.autodesk.com/privacy
.

#### Turn Communication Center On or Off

The Autodesk CAD Manager Control utility turns Communication Center on and off. For example, if you want to prevent Communication Center from sending information to Autodesk, you can turn it off. Information about how to use the utility is available by installing and running the utility, and then clicking Help in the CAD Manager Control Utility window.

#### To install the Communication Center utility (part of the CAD Manager Control utility)

1. Double-click *setup.exe* on the product CD.
2. In the Media Browser, Install tab, click either Multi-Seat Stand-Alone Installation or Network Deployment.
3. Under Install Supplemental Tools, click Autodesk CAD Manager Tools 3.0.
4. Under Autodesk CAD Manager Tools 3.0, click Install.

   After you install the CAD Manager Control utility, you access it from the Start menu (Windows).

# Customize the Update and Announcement Options

After installing your Autodesk product, you configure Communication Center with the Welcome wizard to specify the information you want sent to you.

- **Country/Region.** Specifies your country/region so that Communication Center can provide information that is designed specifically for your location.
- **Update Frequency.** Specifies how often you want Communication Center to synchronize with Autodesk servers.
- **Balloon Notification.** Turns on balloon notification so that Communication Center balloon messages are displayed above the status bar when a new announcement is received. If the balloon notifications have been turned off in the tray settings, the Balloon Notification setting in the Communication Center is ignored.
- **Channels.** Specifies the information that you want displayed in Communication Center.

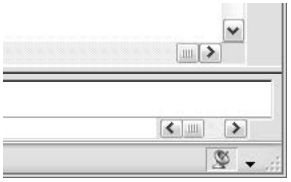
#### To customize Communication Center options

1. Click the Communication Center icon located in the tray on the right side of the status bar.
2. In the Communication Center window, click Settings.
3. In the Configuration Settings dialog box, choose the settings and options that you want to use. Click Apply.
4. Click OK to close the Configuration Settings dialog box. Then, close the Communication Center window.

# Receive New Information Notifications

Whenever new information is available, Communication Center notifies you by displaying a balloon message above the status bar.

Click the balloon message to open the Communication Center window. If you prefer to be notified by the Communication Center icon, you can turn off Balloon Notification in the Configuration Settings dialog box of Communication Center.



If no Communication Center icon is shown on the status bar, see your network administrator.

**To open the Communication Center window**

- Click the Communication Center icon located in the tray on the right side of the status bar.

# 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

# Part 1
# The User Interface

Chapter 2 Menus, Toolbars, and Tool Palettes
Chapter 3 The Command Window
Chapter 4 DesignCenter
Chapter 5 Customize the Drawing Environment
Chapter 6 Pointing Devices

# Menus, Toolbars, and Tool Palettes

# 2

You can use several menus, shortcut menus, tool palettes, and toolbars for access to frequently used commands, settings, and modes.

## In this chapter


- Tool Palettes
- Toolbars
- The Menu Bar
- Shortcut Menus
- The Object Snap Menu
- Create Your Own Menu and Toolbar Groups

# Tool Palettes

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

## Create and Use Tools from Objects and Images

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

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

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

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 will have the same properties as the original tool on the tool palette.

For block and xref tools, you can choose to be prompted at the command line for a rotation angle (starting from 0) when you click and place the block or

xref. 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 line 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. You can set an auxiliary scale for a block or a hatch tool to override 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.)

#### Scale Blocks Automatically

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.

---

**See also:**


"Control Tool Properties" on page 31
"Customize Tool Palettes" on page 35
"Use Workspaces" on page 88
"Add Content with DesignCenter" on page 67


### To display the Tool Palettes window

■ Click Tools menu ➤ Tool Palettes Window. Alternatively, you can press CTRL+3.

Standard toolbar

### To create a tool from an object in the current drawing

1 In the current drawing, select an object such as a dimension, block, hatch, gradient fill, raster image, xref, or any geometric object.

2 Drag the object to a tool palette and, without releasing the mouse button, move the cursor to the place on the tool palette where you want the tool.

   The black line indicates where the tool will be located.

3 Release the mouse button.

Standard toolbar

TOOLPALETTES

**To use a tool created from a geometric object**

1 On a tool palette, click the geometric object tool you want to use.

2 Use the tool as if you selected the corresponding option from the Draw menu or the corresponding button on the Draw toolbar, following the prompts on the command line.

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

Standard toolbar

TOOLPALETTES

**To use a tool created from a dimension**

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

2 Use the tool as if you selected the corresponding option from the Dimension menu or the corresponding button on the Dimension toolbar, following the prompts on the command line.

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

Standard toolbar

TOOLPALETTES

### To use a tool on a tool flyout

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

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

Standard toolbar

TOOLPALETTES

### To add or remove a tool flyout

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

   **NOTE** If you remove the flyout from a tool but then add the flyout back, the image, name, and description (the tooltip) that are shown on the tool palette for each tool on the flyout will not be accurate. To correct this, you must 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" on page 35.

Standard toolbar

TOOLPALETTES

**To customize a tool flyout**

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

Standard toolbar

TOOLPALETTES

**To use a tool created from a hatch**

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

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

Standard toolbar

TOOLPALETTES

**To use a tool created from a gradient fill**

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

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

Standard toolbar

TOOLPALETTES

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

1. On a tool palette, click the block, xref, or raster image you want to insert into your drawing.
2. Drag the block, xref, or raster image to the appropriate place in the drawing.
3. Release the mouse button to insert the block, xref, or raster image.

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

Standard toolbar

TOOLPALETTES

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

1. In a tool palette, right-click a block or xref tool. Click Properties.
2. In the Tool Properties dialog box, under Insert, click Prompt for Rotation.
3. In the drop-down list, select Yes.
4. Click OK.

   **NOTE** 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 line prompt, you enter **rotate**.

Standard toolbar

TOOLPALETTES

# 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.

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

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

**To create a command tool**

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, right-click any displayed toolbar and select another toolbar from the list.

2. Right-click on the title bar of a tool palette. Click Customize to open the Customize dialog box.

   **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 line indicates where the tool will be located.

4. Release the mouse button.

5. In the Customize dialog box, click Close.

CUSTOMIZE

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

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

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

3 Release the mouse button.

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

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

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

7 Click OK.

CUSTOMIZE

**To use a command tool**

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

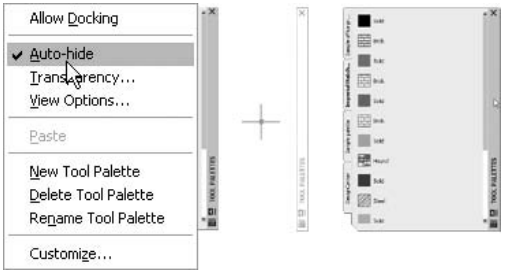
2 Follow any prompts that are shown on the command line.

# Change Tool Palette Settings

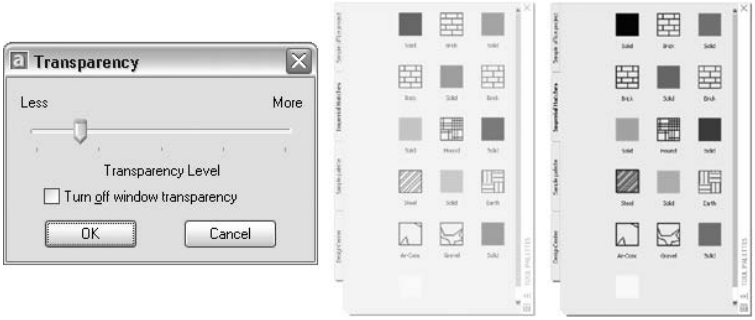
The options and settings for tool palettes are accessible from shortcut menus that are displayed when you right-click in different areas of the Tool Palettes window.

These settings include

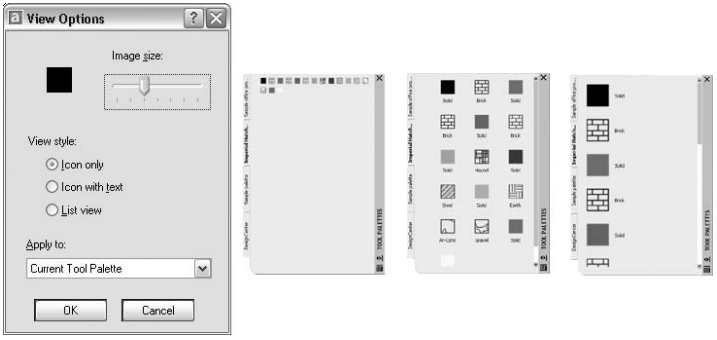
- **Auto-hide.** The Tool Palettes window can automatically roll open or roll away (called *rollover behavior*) when your cursor moves over the title bar on the Tool Palettes window.



- **Transparency.** The Tool Palettes window can be made transparent so it does not obscure objects under it.




- **Views.** The display style and size of the icons in a tool palette can be changed.




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

Tool palette settings are saved with your profile.

**To change the rollover behavior of the Tool Palettes window**

- Click the Auto-Hide button at the bottom of the title bar of the Tool Palettes window.

Rollover behavior is on.

Rollover behavior is off.

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

**To change the transparency of the Tool Palettes window**

1. Right-click the title bar of the Tool Palettes window. 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.

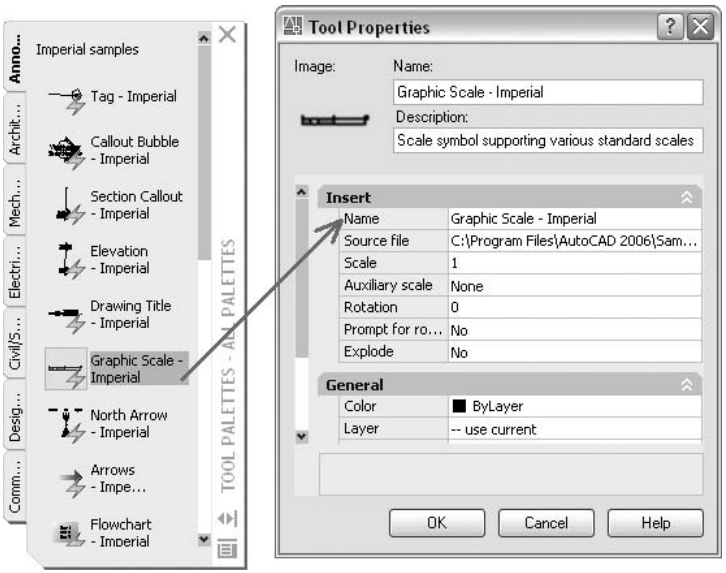
**To change the icon display style in the Tool Palettes window**

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.

# 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.

### 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 selecting Update Tool Image. You must save the drawing before you can update the tool image.

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

### Specify Overrides for Tool Properties

In some cases, you may want to assign specific property overrides to a tool. For example, you may want a hatch to be placed automatically on a

pre-specified layer, regardless of the current layer setting. This feature can save you time and reduce errors by setting properties automatically when creating certain objects.

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

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

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

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

1. On a tool palette, right-click a tool. Click Properties.
2. In the Tool Properties dialog box, use the scroll bar to view all tool properties.

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

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

1. On a tool palette, right-click a tool. Click Properties.
2. In the Tool Properties dialog box, click any property in the list of properties and specify the new value or setting.

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

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

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

Standard toolbar

**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.

Standard toolbar

TOOLPALETTES

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

1. On a tool palette, right-click a tool.
2. Click Properties.
3. In the Tool Properties dialog box, right-click in the Image area. Click Specify Image.

**NOTE** In AutoCAD, this option is available only for geometric object, dimension, and command tools.

4. In the Select Image File dialog box, locate the image file you want to use.
5. Click Open to insert the new image.
6. In the Tool Properties dialog box, click OK.

**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" on page 35.

Standard toolbar

TOOLPALETTES

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

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

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

Standard toolbar

TOOLPALETTES

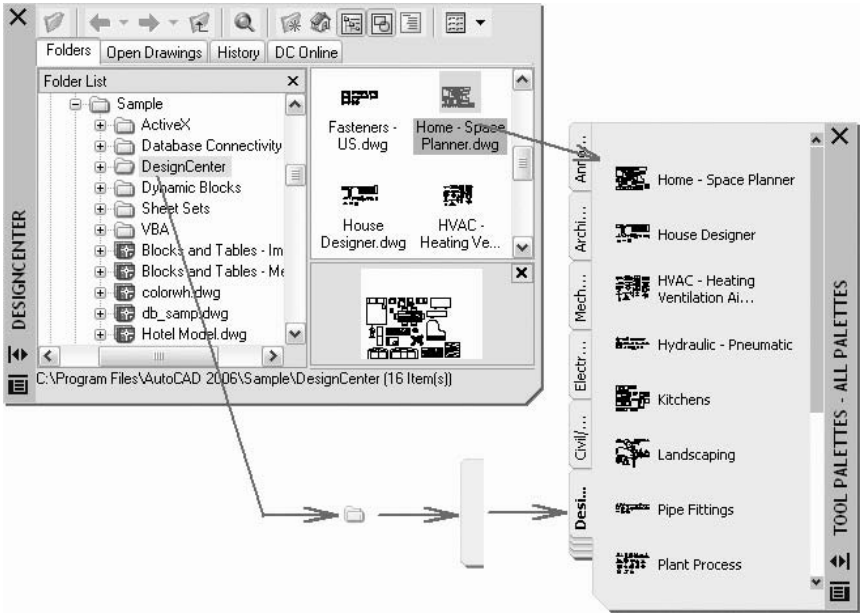
# Customize Tool Palettes

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

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

- Drag any of the following onto your tool palette: geometric objects such as lines, circles, and polylines; dimensions; hatches; gradient fills; blocks; xrefs; raster images.

- Drag drawings, blocks, and hatches from DesignCenter to the tool palette. Drawings that are added to a tool palette are inserted as blocks when dragged into the drawing.
- Using the Customize dialog box, drag commands to a tool palette just as you might add them to a toolbar.
- Use Cut, Copy, and Paste to move or copy tools from one tool palette to another.
- 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.




**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.

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 2006\r16.2\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 from a folder or a drawing

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

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

Standard toolbar

### 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.

Standard toolbar

TOOLPALETTES

**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.

Standard toolbar

TOOLPALETTES

**To sort items in a tool palette**

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

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

Standard toolbar

TOOLPALETTES

# 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 create a group called Hatch Patterns. You can then add all your tool palettes that contain hatch patterns to the Hatch Pattern group. When you set the Hatch Pattern group as the current group, only those tool palettes you've added to the group are displayed.

**To create a tool palette group**

1. Right-click on the title bar of a tool palette. Click Customize.
2. In the Customize dialog box, Tool Palettes tab, 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.

#### To add a tool palette to a tool palette group

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

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

3 Click Close.

#### To remove a tool palette from a tool palette group

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

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

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

3 Click Close.

#### 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 be shown.

You can also specify the tool palette group you want to be displayed using the Customize dialog box. On the Tool Palettes tab, under Palette Groups, right-click the group you want to be shown. Click Set Current.

#### To delete a tool palette group

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

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

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

3 Click Close.

### 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 Right-click on the title bar of a tool palette. Click Customize.

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

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

5 When you are finished, click Close.

### To rename a tool palette group

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

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

3 Enter a new name for the tool palette group.

4 Click Close.

### To rearrange tool palette groups

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

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

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

3 Click Close.

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

### To create a tool palette group within another one

1. Right-click on the title bar of a tool palette. Click Customize.
2. In the Customize dialog box, Tool Palettes tab, under Palette Groups, right-click the tool palette group to which you want to add the new group. Click New Group.
3. Enter a name for the new tool palette group.
4. Click Close.

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

1. Right-click on the title bar of a tool palette. Click Customize.
2. In the Customize dialog box, Tool Palettes tab, 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 change the order of displayed tool palettes within a group

1. Right-click on the title bar of a tool palette. Click Customize.
2. In the Customize dialog box, Tool Palettes tab, under Palette Groups, drag a tool palette to the new location in its tool palette group.
3. Click Close.

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

1. Right-click on the title bar of a tool palette. Click Customize.
2. In the Customize dialog box, Tool Palettes tab, under Tool Palettes, drag a tool palette to the location you want.

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

### To display all tool palettes

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

## Save and Share Tool Palettes

Save and share a tool palette by exporting it or importing it as a tool palette file.

You can save and share a tool palette by exporting it or importing it as a tool palette file. You import and export a tool palette from the Tool Palettes tab of the Customize dialog box. Tool palette files have an *.xtp* file extension.

Tool palettes can be used only in the version of AutoCAD in which they were created. For example, you cannot use a tool palette that was created in AutoCAD 2006 in AutoCAD 2005.

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

Tool palette groups are saved in profiles.

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

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

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**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.

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**See also:**

"Save and Restore Profiles" on page 87

# 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 is displayed.

The Standard toolbar at the top of the drawing area is displayed by default. This toolbar is similar to those found in Microsoft® Office programs. It contains frequently used AutoCAD commands such as , , and , as well as Microsoft Office standard commands such as New, Open, and Save.

### Display or Hide, Dock, and Resize Toolbars

AutoCAD initially displays several toolbars:

- Standard toolbar
- Styles toolbar
- Layers toolbar
- Properties toolbar
- Draw toolbar
- Modify toolbar

You can display or hide these toolbars and additional ones. You can also create your own toolbars.

A toolbar can be *floating* or *docked*. A floating toolbar is located 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. You can move a docked toolbar by dragging it to a new docking location.

**See also:**

**"Customize Toolbars" in the Customization Guide**

#### To display a toolbar

1. Right-click any toolbar and click a toolbar on the shortcut menu.

#### To dock a toolbar

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

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

### To undock a toolbar

1. Position the cursor on the double bars at the end of the toolbar, and hold down the button on your pointing device.
2. Drag the toolbar away from its docked location and release the button.

### To resize a toolbar

1. Position the cursor on the edge of a floating toolbar until the cursor changes to a horizontal or vertical double arrow.
2. Hold down the button and move the cursor until the toolbar is in the shape you want.

### To close a toolbar

1. If the toolbar is docked, undock it.
2. Click the Close button in the upper-right corner of the toolbar.

# The Menu Bar

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

Menus are available from the menu bar at the top of the AutoCAD drawing area.

You can specify menus to display in the program by customizing a CUI file and loading it into the program.

**See also:**

**"Create Pull-Down and Shortcut Menus" in the Customization Guide**

### To use a menu

Choose one of the following methods:

- On the menu bar, click a menu name to display a list of options. On the menu, either click an option or use the DOWN ARROW to move down the list, and then press ENTER.
- Press ALT and press the key for the underlined letter in the menu name; then, press the underlined letter in the option name. For example, to open a new drawing, press ALT and press F to open the File menu; then, press N for New.

# 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, including

- Within the drawing area with or without any objects selected
- Within the drawing area during a command
- Within the text and command windows
- Within areas and on icons in DesignCenter
- Within areas and on text in the In-Place Text Editor
- On a toolbar or tool palette
- On the Model or layout tabs
- On the status bar or the status bar buttons
- In certain dialog boxes

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 (CUI) file. The main CUI file is called acad.cui by default.

**See also:**

**"Create Submenus" in the Customization Guide**

### To display a shortcut menu

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

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

### To turn off shortcut menus in the drawing area

1. Click Tools menu ➤ Options.
2. In the Options dialog box, User Preferences tab, under Windows Standard Behavior, clear Shortcut Menus in Drawing Area.
3. To control Default, Edit, and Command shortcut menus individually, select Shortcut Menus in Drawing Area. Right-Click Customization.
4. In the Right-Click Customization dialog box, under Default Mode or Edit Mode, select one of the following options to control what happens when you right-click in the drawing area and no command is in progress:

   - **Repeat Last Command.** Repeats the last command. Selecting this option turns off the Default and Edit shortcut menus. Right-clicking is the same as pressing ENTER .
   - **Shortcut Menu.** Displays the Default or Edit shortcut menu.

5. Under Command Mode, select one of the following options to determine what happens when you right-click in the drawing area while a command is in progress:

   - **Enter .** Turns off the Command shortcut menu. Right-clicking is the same as pressing ENTER .
   - **Shortcut Menu: Always Enabled .** Displays the Command shortcut menu.
   - **Shortcut Menu: Enabled When Command Options Are Present .** Displays the Command shortcut menu only when options are currently available in the Command prompt. In a Command prompt, options are enclosed in square brackets. If no options are available, right-clicking is the same as pressing ENTER.

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

**To turn on time-sensitive right-click behavior**

1. Click Tools menu ➤ Options.
2. In the Options dialog box, User Preferences tab, under Windows Standard Behavior, click Right-Click Customization.
3. In the Right-Click Customization dialog box, select Turn on Time-Sensitive Right-Click.

   You can specify the duration of the longer click. The default is 250 milliseconds
4. Click Apply & Close.
5. In the Options dialog box, click OK.

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

1. Click the arrow at the right end of the status bar, and then click Tray Settings.
2. 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.
3. If Display Notifications from Services is selected, set a time for a notification to be displayed, or select Display Until Closed.
4. Click OK.

Right-click an empty area on the status bar. Click Tray Settings.

**To control the display of buttons on the status bar**

- Click the arrow at the right end of the status bar and click any button name to change the display.

Items with a check mark next to them are displayed on the status bar.

Right-click an empty area on the status bar. Click a button name.

#### To control the display of coordinates on the status bar

- Click the arrow at the right end of the status bar and click Cursor Coordinate Values.

Items with a check mark next to them are displayed on the status bar.

Right-click an empty area on the status bar. Click Cursor Coordinate Values.

#### 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 line 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.

# The Object Snap Menu

Specify an object snap quickly and conveniently from a shortcut menu.

The object snap menu is displayed at your cursor location when you hold down SHIFT and click the right mouse button or the equivalent button on another pointing device.

The default object snap menu lists object snaps and tracking options. If you want to change the options, you can modify a customization file. The main customization file that's shipped with the product is *acad.cui*.

**See also:**

**"Create Pull-Down and Shortcut Menus" in the Customization Guide**

**To display the object snap menu**

1. Enter any command that prompts you to specify a point. For example, enter **line**.
2. At the From Point prompt, hold down SHIFT and right-click.

   The object snap menu is displayed, and you can click an object snap option.

# Create Your Own Menu and Toolbar Groups

Create your own menu and toolbar groups in a workspace.

You can identify task specific toolbars and menus for each workspace that you create or use.

For more information about how toolbars and menus interact with workspaces, see "Use Workspaces" on page 88 and Customize Workspaces in the *Customization Guide*.

# The Command Window

# 3

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.

## In this chapter

- Enter Commands on the Command Line
- Enter System Variables on the Command Line
- Navigate and Edit Within the Command Window
- Switch Between Dialog Boxes and the Command Line
- Dock, Resize, and Hide 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*.

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 command, you can enter **l**. Abbreviated command names are called *command aliases* and are defined in the *acad.pgp* file.

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

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

### Specify Command Options

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

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

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

### Execute Commands

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

### Repeat and Cancel Commands

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

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

Command: **multiple circle**

To cancel a command in progress, press ESC .

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

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

**"Add Shortcut Keys and Temporary Override Keys" in the Customization Guide**

### To copy a command you have recently used

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

# 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  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  command.  is a read-only system variable that stores the current date. You can display this value, but you cannot change it.

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. At the Command prompt, enter **setvar**.
2. At the Variable Name prompt, enter **?**.
3. At the Enter Variable(s) to List prompt, press ENTER.

# 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
- INS, DEL

- 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 ARROW 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 , 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  command.

**See also:**

"Dock, Resize, and Hide the Command Window" on page 57
"Use Dynamic Input" on page 335

### To display the text window

- From within the drawing area, press F2.

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

**To close the text window**

- From within the text window, press F2.

The text window is closed. You can also close the text window using the standard Windows controls.

**To copy text from the text window to the command line**

1. If the text window is not displayed, press F2 to display it.
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.

   ,

# Switch Between Dialog Boxes and the Command Line

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

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

These system variables also affect the display of dialog boxes:

- controls whether uses a dialog box for attribute value entry.
- displays the name (in English) of the currently active command and transparent command.

- controls whether certain warning dialog boxes are displayed.
- controls the display of dialog boxes used with commands that read and write files. For example, if FILEDIA is set to 1, 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.

# Dock, Resize, and Hide the Command Window

To relocate and resize the command window for your convenience, drag it or use the splitter bar. To hide the command window, click Command Line on the Tools menu.

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

You can resize the 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.

Undock the command window by dragging it away from the docking region. You can move the floating command window anywhere on the screen and resize its width and height with the pointing device.

Dock the command window by dragging it until it is over the top or bottom docking region of the AutoCAD window.

Hide the command line by clicking the Tools menu ➤ Command Line (or 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 display the command line in those

instances. To display the command line when it's hidden, click the Tools menu ➤ Command Line (or press CTRL+9).

**NOTE** For information about display options (such as auto-hide or transparency) for dockable windows, see "Control the Size, Location, and Appearance of Dockable Windows" in the topic "Set Interface Options" on page 80.

### To float the command window

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

### To make the floating command window transparent

1. Right-click in the floating command window. Click Transparency.
2. In the Transparency dialog box, move the slider to the left to make the command window less transparent and to the right to make it more transparent.

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

### To dock the command window

- Click the title bar and drag the command window until it is over the top or bottom docking region of the AutoCAD window.

### 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

- On the Tools menu, click 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 or click the Tools menu ➤ Command Line.

# DesignCenter

# 4

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.

## In this chapter


- Overview of DesignCenter
- Understand the DesignCenter Window
- Access Content with DesignCenter
- Add Content with DesignCenter
- Retrieve Content from the Web with DesignCenter Online

# 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

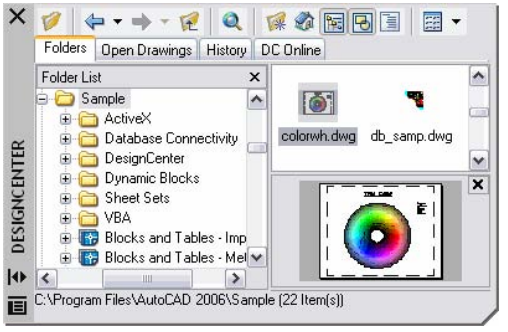
# Understand the DesignCenter Window

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.

- Resize DesignCenter by dragging the bar between the content area and the tree view or by dragging an edge as you would any other window.
- Dock DesignCenter by dragging it over the right or left docking region of the AutoCAD® 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.
- Change the automatic rollover behavior of DesignCenter by clicking the Auto-hide button on the DesignCenter title bar.

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.

Several options are available on the shortcut menu that is displayed when you right-click on the DesignCenter title bar.

### 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  command. The same navigation and display options are available on a shortcut menu. Right-click in the DesignCenter content area.

### To change the DesignCenter rollover behavior

1. If DesignCenter is not already open, click Tools menu ➤ 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.

Standard toolbar

**To prevent DesignCenter from docking**

1. If DesignCenter is not already open, click Tools menu ➤ DesignCenter.
2. Click and hold on the DesignCenter title bar. Press CTRL as you move your mouse.

Standard toolbar

**To display and hide the DesignCenter tree view**

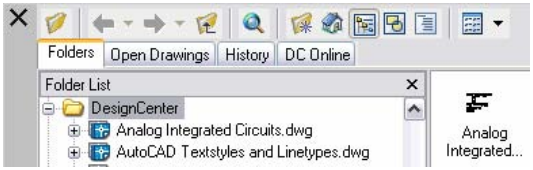
1. If DesignCenter is not already open, click Tools menu ➤ DesignCenter.
2. On the DesignCenter toolbar, click Tree View Toggle.

Standard toolbar

Right-click the content area background and click Tree.

# 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, 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.

#### 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. If DesignCenter is not already open, click Tools menu ➤ 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.

Standard toolbar

**To change the folder of the Home button in DesignCenter**

1. If DesignCenter is not already open, click Tools menu ➤ 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.

Standard toolbar

**To add items to the *Favorites* folder in DesignCenter**

1. If DesignCenter is not already open, click Tools menu ➤ DesignCenter.
2. Right-click the item in the DesignCenter tree view or content area. Click Add to Favorites.

Standard toolbar

**To display the contents of the Favorites folder in DesignCenter**

1. If DesignCenter is not already open, click Tools menu ➤ 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.

Standard toolbar

**To organize your DesignCenter Favorites folder**

1. If DesignCenter is not already open, click Tools menu ➤ 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.

Standard toolbar

# 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. If DesignCenter is not already open, click Tools menu ➤ 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.

Standard toolbar

**To load the content area from the DesignCenter Search dialog box**

1 If DesignCenter is not already open, click Tools menu ➤ 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.

Standard toolbar

**To load the content area of DesignCenter with a symbol library**

1 If DesignCenter is not already open, click Tools menu ➤ 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.

---

Standard toolbar

**To load the content area of DesignCenter with hatch patterns**

1 If DesignCenter is not already open, click Tools menu ➤ 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.

Standard toolbar

**To open a drawing from DesignCenter**

1 If DesignCenter is not already open, click Tools menu ➤ 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.)

Standard toolbar

**To update a block definition with DesignCenter**

1. If DesignCenter is not already open, click Tools menu ➤ 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.

Standard toolbar

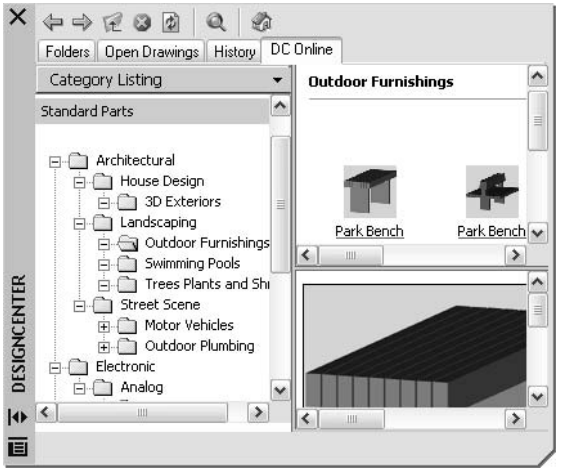
# Retrieve Content from the Web with DesignCenter Online

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

## Overview of DesignCenter Online

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

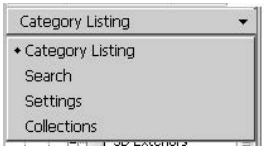
To access DesignCenter Online, click the DC Online tab in DesignCenter. Once the DesignCenter Online window is open, you can browse, search, and download content to use in your drawing.




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

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

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




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

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

### DesignCenter Online Privacy

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

The following information is sent to Autodesk:

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

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

### Turn the DC Online Tab On or Off

The CAD Manager Control utility turns the DC Online tab in DesignCenter on and off. Information about how to use the utility is available by installing and running the utility, and then clicking Help in the CAD Manager Control Utility window.

#### To install the CAD Manager Control utility

1. Insert the CD or DVD, and double-click *setup.exe*.
2. In the Media Browser, Network Deployment tab, under Install Supplemental Tools, click Autodesk CAD Manager Tools 2.0.
3. Under Autodesk CAD Manager Tools 2.0, click Install.

After you install the CAD Manager Control utility, you access it from the Start menu (Windows). Information about how to use the utility is available in Help on the CAD Manager Control Utility window.

# 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.

#### 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.

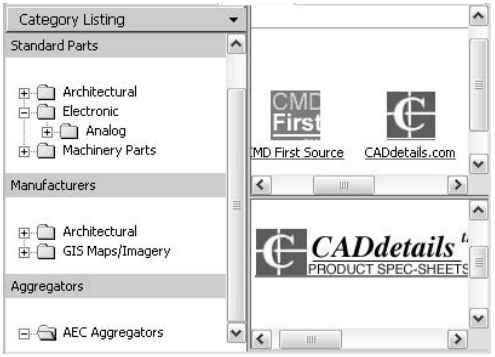
The category folders are displayed in the left pane of the window.

# Retrieve Content from the Web

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

#### Browse for Content

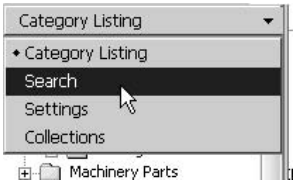
When you use the Category Listing view, you can click the folders in the left pane to view their contents. These folders may contain other folders.




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.

#### 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.

# Customize the Drawing Environment

# 5

You can change many window and drawing environment settings in the Options dialog box, when starting the program, or on the fly. 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.

## In this chapter


- Set Interface Options
- Customize Startup
- Save and Restore Profiles
- Use Workspaces

# Set Interface Options

You can change the appearance of elements in the drawing area and specify other aspects of your working environment, for example, how often the drawing is automatically saved.

In the Options dialog box, you can change many of the settings that affect the interface and the drawing environment. Experiment until you find the best environment for your needs.

- **Automatic Save (Open and Save tab)**. Saves your drawing at specified time intervals. To use this option, in the Options dialog box, Open and Save tab, select Automatic Save and enter the interval in minutes.
- **Search Path (Files tab)**. Sets the search path the program uses to find drawing support files such as text fonts, drawings, linetypes, and hatch patterns.

### Change the Appearance of Elements in the Drawing Area

Some settings in the Options dialog box affect the appearance of the drawing area, for example:

- **Color (Display tab)**. Specifies the background colors used in the layout and Model tabs and the color used for prompts and crosshairs.
- **Font (Display tab)**. Changes the fonts used in the application window and in the text window. This setting does not affect the text in your drawings.

### Clear the Screen

To expand the drawing display area, click View menu ➤ Clean Screen to display only the menu bar, status bar, and command window. A check mark is displayed next to the option. Click Clean Screen again to restore the previous setup.

### Calculate Mathematical Expressions in a Dialog Box

You can enter and evaluate mathematical expressions in a dialog box using the following format: *=expression<END>*.

---

**NOTE** To evaluate expressions in a dialog box, make sure the system variable, CALCINPUT, is set to 1.

---

### Control the Size, Location, and Appearance of Dockable Windows

Windows that can dock or float, such as the Properties palette, tool palette windows, and DesignCenter, can be resized and placed where you want them in the drawing area.

- **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**. Right-click the title bar. Click Allow Docking. A check mark is displayed next to the option.

  To dock the window, drag it over the right or left edge of the application window until it snaps into the docked position. You can also double-click the title bar to dock the window.

  To undock the window, drag the double lines at the top of the docked window away from the docking region. You can press CTRL to prevent docking while you drag.
- **Auto-hide**. Click the Auto-hide button to automatically roll open or roll up the window when your cursor moves over its title bar. This option is also available on a shortcut menu that is displayed when you right-click the title bar.
- **Transparency**. Right-click the title bar. Click Transparency to make the window transparent so that it does not obscure objects under it. This option is not available for all windows.

### Control the Display, Location, and Size 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. Undock a toolbar by clicking an empty area on the toolbar and dragging it into the drawing area. A floating toolbar is located anywhere in 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 toolbars and dockable windows arranged the way you want them, you can lock their position, whether they are docked or floating. Locked toolbars and windows can still be opened and closed and items can be added and deleted. To unlock them temporarily, hold down CTRL .

### Control View Transitions

You can control whether view transitions appear smooth or instantaneous (). The default is a smooth transition from one view to another. The view transitions settings affect the change of view when you pan or zoom or change from one view to another, including a named view.

### Set the Appearance of Tooltips

Tooltips provide information for many of the drafting aids. The  system variable turns them on and off. The  system variable can combine the display of information into one tooltip. You can set the general appearance of tooltips in the Tooltip Appearance Dialog Box

### To set options

1. Click Tools menu ➤ Options.
2. In the Options dialog box, click the tab you want.
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 Tools menu ➤ Options.
2. In the Options dialog box, Display tab, click Colors.
3. In the Color Options dialog box, select the element you want to change by clicking the images for the Model tab or layout tabs.

   As you click areas of the images, the selection is displayed in the Window Element list. You can also change an attribute by selecting it from the Window Element list.
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 colors specified by the Windows Control Panel, click Default All.
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 font displayed in the command window**

1 Click Tools menu ➤ 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 Click OK to close the Options dialog box.

**To lock the position and size of toolbars and dockable windows**

- Click Window menu ➤ Lock Location.
- Place a check mark next to one or more of the following options, or click All ➤ Locked:
  - Floating Toolbars
  - Docked Toolbars
  - Floating Windows
  - Docked Windows

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

Right-click the lock icon in the system tray. Place a check mark next to one or more options or click All ➤ Locked.

**To evaluate mathematical expressions in a dialog box**

---

**NOTE** To evaluate expressions in a dialog box, the system variable, CALCINPUT, must be set to 1.

---

- In a dialog box, in an area where you can enter a numeric value, enter a mathematical expression in the following format: =*expression* (for example, =5+3).

- Click END on the keyboard.
- The expression is evaluated and its value is displayed.

CALCINPUT

**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 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 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.

# Customize Startup

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

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

Command line switches are parameters you can add to the *acad.exe* command line associated with a Microsoft® Windows® shortcut icon or the Windows Run dialog box. You can include several switches within a single command line. Valid switches are listed in the following table.

| | | |
|---|---|---|
| **/b** | Script name | Designates a script to run after you start the program (b stands for batch process). Scripts can be used to set up drawing parameters in |

| | | |
|---|---|---|
| | | a new drawing file. An SCR file type is assumed. |
| **/t** | Template file name | Creates a new drawing based on a template or prototype drawing. A DWT file type is assumed. |
| **/c** | Configuration folder | Specifies the path for the hardware configuration file that you want to use. You can specify a directory or a particular file. A CFG file type is assumed.<br>If you don't set the /c switch, the executable directory is searched and the ACADCFGW or ACADCFG environment variable is used as a way to define the configuration file and directory location. |
| **/v** | View name | Designates a particular view of the drawing for display at startup. |
| **/ld** | ARX or DBX application | Loads a specified ARX or DBX application. Use the following format:<br><path>\<filename>.ARX<br>If the path or file name contains spaces, then the path or file name should be wrapped in double quotes. If no path information is included, the program search path is used. |
| **/s** | Support folders | Designates support folders other than the current folder. Drawing support files include text fonts, menus, AutoLISP files, linetypes, and hatch patterns. The maximum number of folders you can specify in the path is 15. Each folder name is delimited by semicolons. |
| **/r** | Default system pointing device | Restores the default system pointing device. It creates a new configuration file (*acad.cfg*) and renames the previous *acad.cfg* file to *acad.bak*. |
| **/nologo** | No AutoCAD logo screen | Starts the program without first displaying the logo screen. |
| **/p** | User-defined registry profile for starting the program | Specifies a user-defined registry profile for starting the program. The selected profile is in effect only for the current session of the program, unless you make another profile current in the Options dialog box during that session.<br>You create or import profiles on the Profiles tab in the Options dialog box. With the /p switch, you can specify only those profiles that |

| | | |
|---|---|---|
| | | are listed in the Options dialog box. If the profile does not exist, the current profile is used. |
| **/nossm** | No Sheet Set Manager window | Suppresses the display of the Sheet Set Manager window on startup. |
| **/set** | Sheet set | Loads the named sheet set on startup. Use the following format:<br><path>\<sheet set data file>.DST |

The syntax for using command line switches is

"*drive*:*pathname**acad.exe*" ["*drawing name*"] [/*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 2006* with the drawing template *arch1.dwt*, restores a named view PLAN1, and executes a script file *startup.scr*.

"d:\ AutoCAD 2006*acad.exe*"/t "d:\AutoCAD 2006\template\arch1" /v "plan1" /b "startup"

The environment settings are resolved in the following way:

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

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

**To start the program with a command line switch**

1. Right-click the program icon on the Windows desktop. Click Properties.
2. In the AutoCAD Properties dialog box, Shortcut tab, in the Target box, edit the parameters for the switch using the following syntax:

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

Valid switches are as follows:

**/b** Script name (*b* stands for batch process)

**/t** Template file name

**/c** Configuration folder

**/v** View name

**/s** Support folders

**/r** Default system pointing device

**/nologo** No AutoCAD logo screen

**/p** User-defined registry profile

**/nossm** No Sheet Set Manager window

**/set** Sheet Set name

For example, enter "**d:**\AutoCAD 2006\ *acad.exe* " **/t** "**d:**\AutoCAD 2006**\template\arch1**" **/v** "**plan1**" **/b** "**startup**"

3 Click OK.

# Save and Restore Profiles

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

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

The profile information is stored in the system registry and can be saved to a text file (an ARG file). The program organizes essential data and maintains changes in the registry as necessary.

Once you save a profile, you can export or import the ARG file to and from different computers. If you make changes to your current profile during a work session and you want to save those changes in the ARG file, you must export the profile. When you export the profile with the current profile name, the ARG file is updated with the new settings. You can import the profile again to update your profile settings.

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

### To make a profile current

1 Click Tools menu ➤ Options.

2 In the Options dialog box, Profiles tab, select the profile you want to make current.

3 Click Set Current.

4 Click OK.

**To save a profile**

1 Click Tools menu ➤ Options.

2 In the Options dialog box, Profiles tab, click Add to List.

3 In the Add Profiles dialog box, enter a Profile name and Description.

4 Click Apply & Close to record the current option settings in the system registry and close the dialog box.

5 Click OK.

**To make a profile current before starting the program**

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

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

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

3 Click OK.

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

# Use Workspaces

Workspaces are sets 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. When you use a workspace, the menus, toolbars, and dockable windows display only those options that are relative to the workspace.

For example, if you typically do 2D drafting, you can use a 2D drafting workspace to streamline those drafting tasks. Or if your work primarily involves publishing drawings, you can create a workspace that contains publishing-related toolbars, menus, and dockable windows. You can also modify workspaces, switch between workspaces as you need them, change

workspace settings, or use the default workspace that is shipped with the product.

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.

Workspaces help you to

- Streamline common tasks
- Engage in best practices for drawing tasks and workflow
- Customize your drawing environment

### Workspaces and Profiles

While workspaces change the display of your drawing environment, much like profiles do, workspaces are not the same as profiles.

*Workspaces* control the display of menus, toolbars, and dockable windows in the drawing area. When you use or switch a workspace, you change the display of your drawing area. You can easily switch to another workspace within a drawing session. You manage your workspaces from the Customize User Interface dialog box.

*Profiles* collect many of your user options, drafting settings, paths, and values. Profiles are updated each time you make a change to an option, setting, or other value. If you have previously saved display settings to different profiles, you can use workspaces to switch between display environments. You can manage most of your profiles from the Options dialog box.

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

For more information about profiles, see "Save and Restore Profiles" on page 87.

### Create or Change a Workspace

You can create your own workspaces and modify the default workspace. You can create a workspace that contains only those toolbars and menus that you want. To create or change a workspace, you use the Customize User Interface dialog box to set the workspace environment. You can also create a workspace by rearranging toolbars and windows, and then saving the current workspace

(from the Workspaces toolbar, the Window menu, or using the WORKSPACE command).

**NOTE** For more information about creating or modifying workspaces, see "Customize Workspaces" in the *Customization Guide*.

### Switch Workspaces

If you use the product for a variety of tasks, you might have several workspaces. You can switch to another workspace whenever you need to work on a different task.

### Change Workspace Settings

You can change workspace settings, such as which workspace you want to display or whether or not to save your configuration when you switch workspaces. You change these settings in the Workspace Settings dialog box.

### Select a Sample Workspace

You can use the sample workspaces that are shipped with the product. These predefined workspaces demonstrate how you might use a workspace to streamline your work tasks. You can also modify these sample workspaces.

You can find the sample workspaces in the following location:

*C:\Documents and Settings\<user name>\Application Data\Autodesk\AutoCAD 2006\R16.2\<product language>\Support\acadSampleWorkspaces.CUI*

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

### To switch workspaces

1. Click Window menu ➤ Workspaces.
2. From the list of workspaces, select the workspace you want to switch to.

### To change workspace settings

1. Click Window menu ➤ Workspaces ➤ Workspace Settings.
2. In the Workspace Settings dialog box, change workspace settings as needed.
3. Click OK.

**To save a workspace**

1. In the drawing area, arrange a drawing environment by moving, deleting, or displaying those toolbars and dockable windows you'd like to preserve for future use.
2. Click Window menu ➤ Workspaces ➤ Save Current As.
3. In the Save Workspace dialog box, enter a name for the new workspace or select a name from the drop-down list.
4. Click Save.

# Pointing Devices

# 6

You can control this program with a pointing device such as a mouse, digitizing puck, or stylus. A pointing device may have a number of buttons. The first 10 buttons are automatically assigned by the program, but you can reassign all but button 1, the pick button, by modifying the menu file. You can change how mouse buttons function by choosing the Mouse icon in the Windows Control Panel.

## In this chapter


- Pointing Device Buttons
- Digitizing Tablets

# Pointing Device Buttons

The first 10 pointing device buttons are automatically assigned; you can reassign all except button 1, the pick button.

On a two-button mouse, the left button is the pick button used to

- Specify locations
- Select objects for editing
- Choose menu options and dialog box buttons and fields

The operation of the right button on a mouse depends on context; it can be used to

- End a command in progress
- Display a shortcut menu
- Display the Object Snap menu
- Display the Toolbars dialog box

Right-click operation can be modified in the Options dialog box (). The operation of additional buttons on a pointing device is defined in the menu file.

### The Wheel Mouse

The wheel mouse has a small wheel between the buttons. The left and right buttons behave the same as they do on a standard mouse. You can rotate the wheel by discrete values. You can use the wheel to zoom and pan in your drawing without using any commands.

By default, the zoom factor is set to 10 percent; each increment in the wheel rotation changes the zoom level by 10 percent. The  system variable controls the incremental change, whether forward or backward. The higher the number, the larger the change.

The following table lists the wheel mouse actions supported in this program.

| To... | Do this... |
|---|---|
| Zoom in or out | Rotate the wheel forward to zoom in, backward to zoom out |
| Zoom to drawing extents | Double-click the wheel button |

| To... | Do this... |
|---|---|
| Pan | Hold down the wheel button and drag the mouse |
| Pan (joystick) | Hold down CTRL and the wheel button, and drag the mouse |
| Display the Object Snap menu | With the MBUTTONPAN system variable set to 0, click the wheel button |

**See also:**

**"Customize Mouse Buttons" in the Customization Guide**

### To practice using several features of the mouse

1. Move your mouse and notice that the pointer on the screen changes from crosshairs while the pointer is in the drawing area, to an arrow when it's not over the drawing area, and to an I-beam when it's in the text window.
2. As you continue to move the mouse, notice that the numbers in the coordinate display on the status bar change. These numbers indicate the exact location, or coordinate, of the crosshairs on the screen. Click in the coordinate display to turn it off. Notice that the coordinates are updated only when you click in the drawing area.
3. Find the Snap button on the status bar and click it with the pick button on your mouse (usually the left button). Notice that the button darkens to indicate that Snap mode has been turned on.
4. Move the pointer around the screen and observe that it seems to adhere, or "snap," to points at predetermined and equivalent intervals on the screen. You can change the size of these intervals.
5. Click the Snap button again to turn Snap mode off.
6. Move the pointer over the Standard toolbar at the top of the drawing area. As you leave the pointer over a button for a few moments, notice a pop-up label, called a tooltip, that identifies the button.
7. Move the pointer over double bars at the end of a toolbar. Then, as you hold down the pick button, drag the toolbar around the screen to reposition it.
8. Dock the toolbar by dragging it to a docking location at the top, bottom, or either side of the drawing area. When the outline of the toolbar appears in the docking area, release the pick button.

**To turn off all 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. Click OK to record the current options settings in the system registry and close the Options dialog box.

**To turn off shortcut menus individually**

1. Click Tools menu ➤ Options.
2. In the Options dialog box, User Preferences tab, select Shortcut Menus in Drawing Area, and then click Right-Click Customization.
3. In the Right-Click Customization dialog box, under Default Mode, Edit Mode, or Command Mode, select the options to control what happens when you right-click in the drawing area.
4. Click Apply & Close to close the dialog box.
5. Click OK to record the current options settings in the system registry and close the Options dialog box.

# Digitizing Tablets

You can use the puck or stylus of a digitizing tablet as a pointing device, or you can trace a paper drawing into a file.

A digitizing tablet, or digitizer, is a peripheral device that can be used to trace paper drawings into a drawing file or to choose commands from a digitizing tablet overlay. With the Wintab driver, the tablet pointer can also be used instead of a mouse as a system pointer to choose menu items and drawing objects or to interact with the operating system. The tablet pointer can be a *puck* or a *stylus*.

The digitizing tablet must first be configured and then can optionally be calibrated.

- When the tablet is *configured*, portions of the tablet surface are designated as menu areas and a screen-pointing area.
- When the tablet is *calibrated*, it can be used to trace geometry from an existing paper drawing or photograph into a drawing.

You can easily switch between using the tablet uncalibrated as a system pointer (Tablet mode off) or calibrated for digitizing a drawing (Tablet mode on). Click the Tablet button on the status bar. If a screen-pointing area has been designated, a Float button is displayed on the status bar and toggles the screen-pointing area on and off.

---

**NOTE** A sample digitizing tablet overlay, *tablet.dwg*, is included in the *Sample* folder, which is located in the product installation directory.

---

**See also:**

**"Create Tablet Menus" in the Customization Guide**

# Part 2
# Start, Organize, and Save a Drawing

Chapter 7 Start a Drawing

Chapter 8 Open or Save a Drawing

Chapter 9 Repair, Restore, or Recover Drawing Files

Chapter 10 Maintain Standards in Drawings

# Start a Drawing

# 7

You can start a new drawing several ways, including starting from scratch or using a template file. In each case, you can choose the units of measurement and other unit format conventions.

## In this chapter


- Start a Drawing from Scratch
- Use a Wizard to Start a Drawing
- Use a Template File to Start a Drawing
- Specify Units and Unit Formats
- Add Identifying Information to Drawings

# Start a Drawing from Scratch

A quick way to begin a new drawing is to start from scratch, which starts a drawing that uses settings from a default drawing template file.

You can start from scratch from the Create New Drawing dialog box, from the Select Template dialog box, or without using a dialog box at all. In each case, you use either the  command or the  command.

### Use the Create New Drawing Dialog Box

The Create New Drawing dialog box is displayed under the following conditions:

- You set the  system variable to 1 (on).
- You set the  system variable to 1 (on).
- You click New on the File menu or Qnew on the Standard toolbar when no drawing template file is specified in the Options dialog box.

The Create New Drawing dialog box provides several methods for starting a new drawing.

When you use Start from Scratch, you can specify either imperial or metric units for the new drawing. The setting you select determines default values used for many system variables controlling text, dimensions, grid, snap, and the default linetype and hatch pattern file.

- **Imperial .** Creates a new drawing based on the imperial measurement system. The drawing uses internal default values, and the default grid display boundary, called the *grid limits*, is 12 x 9 inches.
- **Metric .** Creates a new drawing based on the metric measurement system. The drawing uses internal default values, and the default grid display boundary is 420 x 290 millimeters.

### Use the Select Template Dialog Box

The Select Template dialog box is displayed under the following conditions:

- You set the  system variable to 0 (off).
- You set the  system variable to 1 (on).
- You click New on the File menu or, by default, Qnew on the Standard toolbar.

At the bottom-right corner of the Select Template dialog box, there is an Open button with an arrow button next to it. If you click the arrow button, you can choose between two internal default drawing templates, metric or imperial.

#### Use a Default Drawing Template File

The fastest way to begin working on a new drawing is to start automatically with a default drawing template file. No dialog box is displayed with this method:

- You set the  system variable to 0 (off).
- You set the  system variable to 1 (on).
- You specify a default drawing template file in the Options dialog box on the Files tab. Click the node labeled "Drawing Template Settings" and specify a path and drawing template file.
- You click Qnew on the Standard toolbar.

From now on, a new drawing is immediately created that is based on the specified default drawing template file.

If no template file is specified in the Options dialog box, QNEW displays the Select Template dialog box.

#### To start a drawing from scratch with the Create Drawing dialog box

1. If necessary, set the STARTUP system variable to 1 and the FILEDIA system variable to 1. On the command line, enter **startup** and **1**; then enter **filedia** and **1**.
2. Click File menu ➤ New.
3. In the Create New Drawing dialog box, click Start from Scratch.
4. Under Default Settings, click either Imperial or Metric.

   The new drawing starts as *drawing1.dwg*. The default drawing name changes to reflect the number of new drawings that have been started. For example, if you start another drawing, the default drawing name is *drawing2.dwg*.

#### To start a drawing from scratch from a default drawing template file

1. If necessary, set the STARTUP system variable to 0 and the FILEDIA system variable to 1. On the command line, enter **startup** and **0**; then enter **filedia** and **1**.
2. Click Tools menu ➤ Options.

3. In the Options dialog box, Files tab, in the list of nodes, click the plus sign (+) next to Template Settings. Click the plus sign (+) next to Drawing Template File Location, and specify a folder path.
4. Click the plus sign (+) next to Default Template File Name for QNEW, and specify a drawing template file name.
5. Click OK.
6. On the Standard toolbar, click Qnew.

   The new drawing starts as *drawing1.dwg*. The default drawing name changes to reflect the number of new drawings that have been started. For example, if you start another drawing, the default drawing name is *drawing2.dwg*.

Standard toolbar

# Use a Wizard to Start a Drawing

A setup wizard establishes basic drawing settings step by step.

You have two wizard options to help you set up a drawing:

- **Quick Setup Wizard .** Sets units of measurement, precision of displayed units, and grid limits.
- **Advanced Setup Wizard .** Sets units of measurement, precision of displayed units, and grid limits. Also establishes angle settings such as units of measurement style, precision, direction, and orientation.

These wizards are available in the Create New Drawing dialog box.

### To begin a new drawing using a wizard

1. If necessary, set the STARTUP system variable to 1 and the FILEDIA system variable to 1.
2. Click File menu ➤ New.
3. In the Create New Drawing dialog box, click Use a Wizard.
4. Click Quick Setup or Advanced Setup.
5. Complete the wizard pages using the Next and Back buttons to move forward and backward.

6 On the last page, click Finish.

# Use a Template File to Start a Drawing

A drawing template file contains standard settings. Select one of the template files supplied, or create your own template files.

Drawing template files have a *.dwt* file extension.

When you create a new drawing based on an existing template file and make changes, the changes in the new drawing do not affect the template file. You can use one of the template files supplied with the program, or you can create your own template files.

### 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 template file instead of specifying the conventions and default settings each time you start. Conventions and settings commonly stored in template files include

- Unit type and precision
- Title blocks, borders, and logos
- Layer names
- Snap, Grid, and Ortho settings
- Grid limits
- Dimension styles
- Text styles
- Linetypes

By default, drawing template files are stored in the *template* folder, where they are easily accessible.

### Recover the Default Drawing Template File

If the settings in the drawing template file *acad.dwt* or *acadiso.dwt* have been changed from the original defaults, you can reset them by starting a new drawing with no template and then saving the drawing as a drawing template file, replacing *acad.dwt* or *acadiso.dwt*.

You can start a new drawing with the original defaults by using NEW to display the Select Template dialog box. To do this, click the arrow next to the Open button and then click one of the "no template" options from the list.

### To start a drawing by selecting a template file

1. Click File menu ➤ New.
2. In the Select Template dialog box, select a template from the list.
3. Click Open.

   The drawing is opened as *drawing1.dwg*. The default drawing name changes to reflect the number of new drawings that have been opened. For example, if you open another drawing from a template, the default drawing name is *drawing2.dwg*.

   If you want 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 File menu ➤ Open.
2. In the Select File dialog box, select the file you want to use as a template.
3. Click OK.
4. If you want to delete the existing file contents, click Modify menu ➤ Erase.
5. At the Select Objects prompt, enter **all**, and then select the border and title block (if you want to remove them) and enter **r** (Remove).
6. Click File menu ➤ Save As.
7. In the Save Drawing As dialog box, under Files of Type, select the Drawing Template file type.

   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.
8. In the File Name box, enter a name for the template.
9. Click Save.
10. Enter a description of the template.
11. Click OK.

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

Standard toolbar

**To recover the default drawing template files**

1. Click File menu ➤ New.
2. In the Select Template dialog box, click the arrow next to the Open button. Select one of the options from the list:
   - Open with No Template - Imperial (to restore *acad.dwt)*
   - Open with No Template - Metric (to restore *acadiso.dwt)*

   The drawing opens with the default settings.
3. Click File menu ➤ Save As.
4. In the Save Drawing As dialog box, select the Drawing Template file type. Save the drawing with its original name, either *acad.dwt* for imperial or *acadiso.dwt* for metric.
5. Click Save.

# 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

Every object you create is measured in drawing units. Before you start to draw, you must decide what one drawing unit will represent based on what you plan to draw. Then you create your drawing at actual size with that convention. For example, a distance of one drawing unit typically represents one millimeter, one centimeter, one inch, or one foot in real-world units.

**Convert Drawing Units**

If you start a drawing in one system of measurement (imperial or metric) and then want to switch to the other system, use  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 806

### To set the units format and precision

1. Click Format menu ➤ Units.
2. In the Drawing Units dialog box, under Length, select a unit format and precision.

   The Sample Output area shows an example of the unit format at the current precision.
3. Click OK.

### To convert a drawing from inches to centimeters

1. Click Modify menu ➤ 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.

Modify toolbar

### To convert a drawing from centimeters to inches

1. Click Modify menu ➤ 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.

Modify toolbar

# Set Linear Unit Conventions

Before you start to draw, you set the format and the number of decimal places to be used when you enter and display linear units.

You can choose from several common conventions to represent the display style and the precision of distances and coordinates. Depending on what you specify, you can enter and display in decimal format, fractional format, or other notation. These settings affect

- The Properties palette
- Dynamic input
- The LIST command
- The ID command
- The coordinate display on the status bar
- Several dialog boxes that display coordinates

You can set the unit type and precision in the Drawing Units dialog box, the Quick Setup wizard, or the Advanced Setup wizard.

### Understand Rounding and Precision

When you specify the display precision of units, the values for coordinates and distances are rounded off. However, the internal precision of coordinates and distances is always maintained regardless of the display precision.

For example, if you set the display precision of decimal-format units to 1 (or 0.0), the display of coordinates is rounded to one place after the decimal point. Thus, the coordinates 0.000,1.375 are displayed as 0.0,1.4, but the internal precision is still maintained.

### Enter Distances in Imperial Architectural Format

In architectural format, to enter feet and inches, indicate feet using the prime symbol ('), for example, 72'3. You don't need to enter quotation marks (") to specify inches.

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**NOTE** The units format for creating and listing objects, measuring distances, and displaying coordinate locations is separate from the dimension units setting used in creating dimension values.

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**See also:**

"Set the Scale for Dimensions" on page 806

### To set the units format and precision

1. Click Format menu ➤ Units.
2. In the Drawing Units dialog box, under Length, select a unit format and precision.

   The Sample Output area shows an example of the unit format at the current precision.
3. Click OK.

### To convert a drawing from inches to centimeters

1. If necessary, click the Model tab to make the Model tab current.
2. Click Modify menu ➤ Scale.
3. At the Select Objects prompt, enter **all**.

   All objects in the drawing are selected for scaling.
4. 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.
5. 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.

Modify toolbar

### To convert a drawing from centimeters to inches

1. If necessary, click the Model tab to make the Model tab current.
2. Click Modify menu ➤ Scale.
3. At the Select Objects prompt, enter all.

   All objects in the drawing are selected for scaling.
4. 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.
5. 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.

Modify toolbar

# Set Angular Unit Conventions

Angle conventions include the position of angle 0 and the direction of measurement: clockwise or counterclockwise. You also set the format and the number of decimal places.

You can choose from several common conventions to represent angles in a drawing. You can specify that positive values of angles are measured either clockwise or counterclockwise, and angle 0 can be set to any direction (usually East or North). You can enter angles using grads, radians, or surveyor's units or using degrees, minutes, and seconds.

### Work with Surveyor's Angles

If you use surveyor's angles when specifying polar coordinates, indicate whether the surveyor's angles are in the north, south, east, or west direction. For example, to enter a coordinate relative to the current coordinate for a property line that is 72 feet, 8 inches long with a bearing of 45 degrees north, 20 minutes, 6 seconds east, enter

**@72'8"<n45d20'6"e**

### To set the angle format and precision

1. Click Format menu ➤ Units.

2 In the Drawing Units dialog box, under Angle, select an angle type and precision.

   The Sample Output area shows an example of the angle type at the current precision.

3 Select Direction to specify an angle direction.

   The angle direction controls the point from which angles are measured and the direction in which they're measured. The default starting angle, 0 degrees, is toward 3 o'clock (or east), and positive angle measurement is counterclockwise.

4 Select the options you want to use.

5 Click OK to close each dialog box.

# 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 and "Access Content with DesignCenter" on page 64.

### Use Sheet Set Manager Properties

With Sheet Set Manager, you can assign a sheet title, number, and a description to every sheet in a sheet set. For more information about sheet sets, see "Include Information with Sheets and Sheet Sets" on page 250.

### Display Properties in Fields

You can assign any of the drawing properties to a field in a text object. For more information about fields, see "Use Fields in Text" on page 725.

#### To display drawing properties for the active drawing

1. Click File menu ➤ Drawing Properties.
2. In the Drawing Properties dialog box, click tabs to view the different types of information.

#### To define drawing properties

1. Click File menu ➤ 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.

# Open or Save a Drawing

# 8

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

## In this chapter


- Open a Drawing
- Open Part of a Large Drawing (Partial Load)
- Work with Multiple Open Drawings
- Save a Drawing
- Find a Drawing File
- Specify Search Paths, File Names, and File Locations

# 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 powerful alternatives.

To open a drawing, you can

- Use Open on the File menu to display the Select File dialog box.
- Double-click a drawing in Windows Explorer to launch AutoCAD® and open the drawing. If the program is already running, the drawing opens in the current session rather than in a second session.
- Drag a drawing from Windows Explorer into AutoCAD.

  If you drop a drawing anywhere outside the drawing area—for example, the command line or the blank space next to the toolbars— the drawing is opened. However, if you drag a single drawing into the drawing area of an open drawing, the new drawing is not opened but inserted as a block reference.
- Use DesignCenter to open drawings
- Use the Sheet Set Manager to locate and open the drawings in a sheet set

### Check Drawing Authenticity

If the  system variable is set to On (1), the DWG file format is checked and an alert box is displayed if

- The drawing file format is AutoCAD LT 97 or later or AutoCAD Release 14 or later *and*
- The file was last saved by a program other than AutoCAD LT 97 or later or AutoCAD Release 14 or later

To avoid checking for file authenticity each time you open drawings, clear the Always Show This Dialog Box option when the alert is displayed. Alternatively, you can set  to Off (0).

### Change the Default Drawing Folder

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

**See also:**

"Overview of DesignCenter" on page 62
"Customize Startup" on page 84
"Work with Sheets in a Sheet Set" on page 233

### To open a drawing

1. Click File menu ➤ Open.
2. In the Select File dialog box, select one or more files. Click Open.

   The icons on the left side of the dialog box provide quick access to commonly used files and file locations. To reorder the icons, drag them to a new location. To add, modify, or remove an icon, right-click the icon to display a shortcut menu.

Standard toolbar

### To change the default drawing folder

1. On the command line, enter **rememberfolders**, and then enter **0**.
2. On the Windows desktop, right-click the AutoCAD icon. Click Properties.
3. Click the Shortcut tab.
4. In the Start In box, enter the path of the folder that should be the default when you open or save drawing files.
5. Click OK.

# Open Part of a Large Drawing (Partial Load)

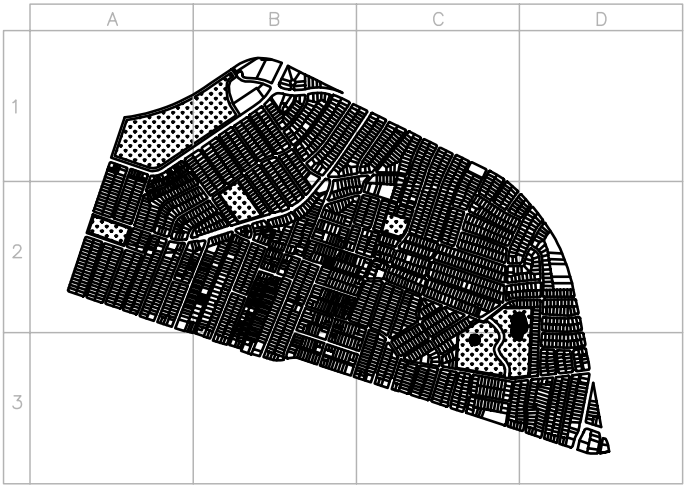
If you work with large drawings, you can improve performance by opening only the view and layer geometry that you want to work with.

If you work with large drawings, you can use the Partial Open option of the OPEN command to select which view and layer *geometry* (graphical objects only) that you want to work with in a drawing. For example, if you load geometry from the EXTENTS view and the SITE layer, everything on the SITE layer that falls within the Extents view is loaded into the drawing.

You can only edit what is loaded into the drawing file, but all the drawing's named objects are available in the partially open drawing. Named objects

include layers, views, blocks, dimension styles, text styles, viewport configurations, layouts, UCSs, and linetypes.

By organizing large drawings into sectored views, you can load and edit only what you need. For example, if you work with a city plan and need to edit only the southeast sector (sector D3 in the illustration), you can load this drawing area by specifying the predefined view. If you need to edit only the city plot numbers, you can load just the geometry on this specific layer.




After a drawing is partially open, you can load additional geometry from a view, selected area, or layer into the drawing by using . The Partial Open option is available only for drawings in AutoCAD 2000 or later format.

**To partially open a drawing**

1. Click File menu ➤ Open.
2. In the Select File dialog box, select a drawing.
3. Click the arrow next to Open. Click Partial Open.
4. In the Partial Open dialog box, select a view; the default view is EXTENTS.

   You can load only geometry from model space views that are saved in the current drawing.
5. Select one or more layers.

If you do not select a layer or layers to load, no layer geometry is loaded into the drawing but all drawing layers exist in the drawing. If no layer geometry is specified to load into the drawing, no geometry is loaded even if the geometry from a view is specified to load. If you draw objects on a layer that is not loaded, you may be drawing on top of existing geometry that is not loaded in the drawing.

**NOTE** Xref-dependent layers are displayed in the Layer Geometry to Load list only if the selected drawing was last saved with the  system variable set to 1. Any layers created in the xref since the xref was loaded into the drawing are not displayed in the Layer Geometry to Load list.

6 If the drawing contains a spatial index, you can select the Use Spatial Index option.

A spatial index is a list that organizes objects based on their location in space. A spatial index is used to locate the portion of the drawing that is read; this minimizes the time required to open the drawing.

7 If the drawing contains xrefs but you don't want to load them, select Unload All Xrefs on Open.

**NOTE** If you partially open a drawing that contains a bound xref, only the portion of the xref that is loaded (defined by the selected view) is bound to the partially open drawing.

8 Click Open.

You can choose to load additional information into the current drawing as long as the drawing is partially open.

### To load additional geometry into a partially open drawing

1 Click File menu ➤ Partial Load.

The Partial Load option is available only if the current drawing is a partially open drawing.

2 In the Partial Load dialog box, select a view or click Pick Window to define a view.

The default view is Extents. You can load only geometry from model space views that are saved in the current drawing.

3 Select one or more layers.

If you do not select a layer or layers to load, no layer geometry is loaded into the drawing, but all drawing layers exist in the drawing. If no layer geometry is specified to load into the drawing, no geometry is loaded even if the geometry from a view is specified to load. If you draw objects

on a layer that is not loaded, you may be drawing on top of existing geometry that is not loaded in the drawing. You cannot unload any geometry that is loaded in the current drawing.

4 Click Open.

# Work with Multiple 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

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

Set the TASKBAR command to 1 to display multiple open drawings as separate items on the Windows taskbar. You can then press Alt+TAB to quickly switch from drawing to drawing. If you prefer to display only the active drawing on the taskbar, set TASKBAR to 0.

### To switch between open drawings

Do one of the following to switch between open drawings:

- Click anywhere in a drawing to make it active.
- Use CTRL + F6 or CTRL + TAB .
- Set the TASKBAR command to 1 to display multiple open drawings as separate items on the Windows taskbar. You can then press Alt+TAB to quickly switch from drawing to drawing.

# 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.

### Save Part of a Drawing File

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

### 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 . 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 IPERCENTSAVE set to 0) before transmitting or archiving a drawing.

### Avoid File Name Issues for International Work

If you share drawing files with companies in other countries and regions, the drawing file names might contain characters that are not available in other languages.

For example, a drawing file created in China might be saved with double-byte characters in the file name. When this drawing file is transmitted to a different language version of Windows, either of the following will occur:

- If Asian language support is turned on, the file name characters are visible in Windows Explorer
- If Asian language support is *not* turned on, the file name characters appear as a series of question marks in Windows Explorer

In *either* case, you will not be able to open the drawing file in AutoCAD LT unless you rename it. This is also true for special characters in file names of European and other languages.

To avoid these issues when your language uses double-byte characters or special characters, do not use high ASCII values (values of 80 hexadecimal and above) when choosing a file name.

**See also:**


**"Save Drawings to Previous Drawing File Formats"**
**"Export Drawings to Other File Formats"**
**"Create Drawing Files for Use as Blocks" on page 456**
**"Add Identifying Information to Drawings" on page 112**
**"Create and Restore Backup Files" on page 130**


### To save a drawing

1. Click File menu ➤ Save.

   If you previously saved and named the drawing, any changes made are saved and the Command prompt is redisplayed. If you have never saved the drawing, the Save Drawing As dialog box is displayed.

2. In the Save Drawing As dialog box under File Name, enter the new drawing name; the file extension is not required. Click Save.

Standard toolbar

### To save a drawing automatically

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Open and Save tab, select Automatic Save.
3. Enter a number in Minutes Between Saves.
4. Click OK.

### To save a backup of the previous version every time the drawing is saved

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Open and Save tab, select Create Backup Copy with Each Save.
3. Click OK.

### To save selected objects into a new drawing file

1. At the Command prompt, enter **wblock**.
2. In the Write Block dialog box, under Objects, select Objects.
3. Under Base Point, click Pick Point.
4. Specify the base point in the drawing area, or enter coordinate values in the *X*, *Y*, and *Z* boxes.
5. Under Objects, click the Select Objects button.
6. Select the objects in the drawing area.
7. Under Objects, select an option to indicate whether you want to retain, convert, or delete the selected objects.
8. In the File Name box, enter the name for the new drawing file.
9. In the Location box, specify the folder for the new drawing file.
10. In the Insert Units box, you can select a different base unit. This option provides a method to scale objects automatically in the new drawing file.
11. Click OK.

    The objects you selected in the drawing are saved in a new drawing file.

### To save a block definition as a new drawing file

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

2 In the Write Block dialog box, select Block.

3 Click in the box next to Block. Select the block definition you want to save.

4 In the File Name box, enter the name for the new drawing file.

5 In the Location box, specify the folder for the new drawing file.

6 In the Insert Units box, you can select a different base unit. This option provides a method to scale objects automatically in the new drawing file.

7 Click OK.

The objects in the specified block definition are saved as objects in a new drawing file. The insertion base point of the block definition will be located at the origin point (0,0,0) in the new drawing.

# 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  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 112
"Access Content with DesignCenter" on page 64
"Work with Sheets in a Sheet Set" on page 233

**To search for files**

1 Click File menu ➤ Open.

2. In the Select File dialog box, click Tools ➤ Find.
3. In the Find dialog box, Name & Location tab, specify a file type, file name, and path.

   You can use wild cards when specifying a file name.
4. On the Date Modified tab, click All Files, or click Find All Files Created or Modified to specify a date filter.

   You can search for drawings modified between a specified range of dates or within a specified number of months or days.
5. Click Find Now.
6. Select one or more files from the search results. Click OK.
7. In the Select File dialog box, click Open.

Standard toolbar

# Specify Search Paths, File Names, 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  system variable stores the location of the *My Documents* folder for the current user.

The working search path for drawing support files lists paths that are valid and exist in the current system directory structure (including system network mapping). Using these options helps improve performance when these files are loaded.

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

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

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

**To change a search path**

1. Click Tools menu ➤ Options.
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 directories until you find the one you want.
5. Select the drive and directory you want to use.
6. Click OK.

# Repair, Restore, or Recover Drawing Files

# 9

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.

## In this chapter


- Repair a Damaged Drawing File
- Create and Restore Backup Files
- Recover from a System Failure

# Repair a Damaged Drawing File

If a drawing file is damaged, you can recover some or all of the data by using commands to find and correct errors.

### Repair and Recovery

When an error occurs, diagnostic information is recorded in the *acad.err* file, which you can use to report a problem.

A drawing file is marked as damaged if corrupted data is detected, or if you request that the drawing be saved after a program failure. If the damage is minor, sometimes you can repair the drawing simply by opening it. Otherwise, you can use the following:

- . Performs an audit on, and attempts to open, any drawing file
- . Finds and corrects errors in the currently open drawing file.

### Example: Auditing Files

Auditing a file generates a description of problems with a drawing file and recommendations for correcting them. As you start the audit, you specify whether you want the program to try to fix the problems it encounters. The report is similar to the following example:

Auditing Header
DXF NAME Current Value Validation Default
PDMODE 990 - 2040
UCSFOLLOW 811 or 0
Error found in auditing header variables
4 Blocks audited
Pass 1 4 objects audited
Pass 2 4 objects audited
Total errors found 2 fixed 2
If you chose not to correct the errors, the last statement changes to

Total errors found 2 fixed 0.

The output from a recovery audit is written to an audit log (ADT) file if the 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 File menu ➤ Drawing Utilities ➤ Recover.
2. In the Select File dialog box, select a file. Click Open.

   After the audit, RECOVER 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).

### 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 File menu ➤ Drawing Utilities ➤ Audit.
2. At the Fix Any Errors Detected? prompt, enter **y** or **n**.

   AUDIT places all objects with errors in the Previous selection set for easy access. The output from the audit is written to an audit log (ADT) file if the AUDITCTL system variable is set to 1 (on).

**NOTE** If the drawing contains errors that AUDIT cannot fix, try using RECOVER. RECOVER repairs any specified DWG file that is not the current drawing file.

**To restore a drawing from a backup file**

1. In Windows Explorer, locate the backup file identified by the *.bak* file extension.
2. Right-click the backup file. Click Rename.
3. Type a new name using the *.dwg* file extension.
4. Open the file as you would open any other drawing file.

# Create and Restore Backup Files

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 131

### 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.

# Recover from a System Failure

A hardware problem, power failure, or software problem can cause this program to terminate unexpectedly. If this happens, you can restore the drawing files that were open.

If the program fails, you can save your current work to a different file. This file uses the format, *DrawingFileName_recover.dwg*, where *DrawingFileName* is the file name of your current drawing.

### Resolve Drawing Files

After a program or system failure, the Drawing Recovery Manager opens the next time you start AutoCAD. Drawing Recovery Manager displays a list of all drawing files that were open, including the following drawing file types:

■ Drawing files (DWG)

■ Drawing template files (DWT)

■ Drawing Standards files (DWS)

**NOTE** Unsaved drawings that are open at the time of an unexpected failure are not tracked by the Drawing Recovery Manager. Be sure to save your work after you begin, and regularly thereafter.

For each drawing, you can open and choose from the following files if they exist:

- *DrawingFileName*_recover.dwg
- *DrawingFileName_a_b_nnnn*.sv$
- *DrawingFileName*.dwg
- *DrawingFileName*.bak

**NOTE** The drawing, backup, and recover files are listed in the order of their time stamps—the time when they were last saved.

Double-click a top-level drawing node listed under Backup Files to display up to four files as listed above. Right-click any node under Backup Files to display shortcut menu options.

If you close the Drawing Recovery window before resolving all affected drawings, you can open Drawing Recovery at a later time with the  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  system variable controls whether the error-reporting feature is available.

#### To open the Drawing Recovery Manager

- Click File menu ➤ Drawing Utilities ➤ 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 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 File menu ➤ Drawing Utilities ➤ 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

- Right-click in the Backup Files area under the last drawing node listed. 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.

# Maintain Standards in Drawings

10

Drawings are easier to interpret if you set standards to enforce consistency. You can set standards for layer names, dimension styles, and other elements; check drawings against these standards; and then change any properties that do not conform.

## In this chapter


- Overview of CAD Standards
- Define Standards
- Check Drawings for Standards Violations
- Translate Layer Names and Properties

# Overview of CAD Standards

You can create a standards file to define common properties in order to maintain consistency throughout your drawing files. Standards define a set of common properties for named objects such as layers and text styles. You or your CAD manager can create, apply, and audit standards in drawings to enforce consistency. Because standards make it easier for others to interpret drawings, standards are particularly useful in collaborative environments, where many individuals contribute to the creation of a drawing.

### Named Objects for Standards-Checking

You can create standards for the following named objects:

- Layers
- Text styles
- Linetypes
- Dimension styles

### Standards File

After you define standards, you save them as a standards file. You can then associate the standards file with one or more drawing files. After you associate a standards file with a drawing, you should periodically check the drawing to make sure it conforms with the standards.

### Sample Drawing and Associated Standards File

With this program, you get a sample drawing and an associated standards file. The drawing file has been deliberately modified to contain a number of nonstandard objects to demonstrate how drawings are audited. The sample files, *MKMPlan.dwg* and *MKMStd.dws*, are installed in the *Sample* folder.

### How a Standards Audit Works

When you check a drawing for standards violations, each named object of a specific type is checked against the standards files associated with the drawing. For example, each layer in the drawing is checked against the layers in the standards file.

A standards audit can uncover two types of problems:

- An object with a nonstandard name is present in the drawing being checked. For example, a layer named WALL is present in the drawing but not in any associated standards files.

- A named object in a drawing matches the name of one in a standards file, but their properties are different. For example, in the drawing the WALL layer is yellow, but the standards file specifies red for the WALL layer.

When you fix objects with nonstandard names, the nonstandard objects are purged from the drawing. Any drawing objects that are associated with the nonstandard object are transferred to a replacement standard object that you specify. For example, you fix a nonstandard layer, WALL, and replace it with the standard ARCH-WALL. In this example, choosing Fix in the Check Standards dialog box transfers all objects from layer WALL to layer ARCH-WALL and then purges layer WALL from the drawing.

#### Standards Plug-Ins

The auditing process uses standards *plug-ins,* applications that define the rules for the properties that are checked for individual named objects. Layers, dimension styles, linetypes, and text styles are each checked against their corresponding plug-ins. You can specify which plug-ins to use when checking a drawing for standards violations. Autodesk or third-party developers may add standards plug-ins for checking additional drawing properties.

All plug-ins check all properties for each named object except for the layer plug-in. The following layer properties are checked when using the layer plug-in:

- Color
- Linetype
- Lineweight
- Plot style mode
- Plot style name (when the PSTYLEMODE system variable is set to 0)

The following layer properties are *not* checked by the layer plug-in:

- On/Off
- Freeze/Thaw
- Lock
- Plot/No Plot

### Standards Settings

A variety of settings that may be useful to CAD managers are available in the CAD Standards Settings dialog box. This dialog box can be accessed by clicking Settings in the Check Standards and Configure Standards dialog boxes.

# Define Standards

To set standards, you create a file that defines properties for layers, dimension styles, linetypes, and text styles, and you save it as a standards file with the *.dws* file name extension.

Depending on how you organize your projects, you may decide to create and associate more than one project-specific standards file with an individual drawing. When you audit the drawing file, conflicts may arise between settings in the standards files. For example, one standards file specifies that the layer WALL is yellow, and another standards file specifies that it is red. In the case of conflicts, the first standards file associated with the drawing takes precedence. If necessary, you can change the order of the standards files to change the precedence.

If you want to audit drawings using just a specific plug-in, you can specify the plug-in when defining your standards file. For example, if recent changes to a drawing are limited to text changes, you may want to audit the drawing using only the layers and text styles plug-ins to save time. By default, all plug-ins are used when auditing drawings for standards violations.

### To create a standards file

1. Click File menu ➤ New.
2. Enter a template file name or press ENTER to continue.
3. In a new drawing, create any layers, dimension styles, linetypes, and text styles that you want to be part of the standards file.
4. Click File menu ➤ Save As.
5. In File Name, enter a name for the standards file.
6. In the Files of Type list, select AutoCAD Drawing Standards (*.dws).

   DWS files must be saved in the current drawing file format. To create a DWS file in a previous drawing file format, save the file in the desired DWG format, and then rename the DWG file using a *.dws* extension.
7. Click Save.

**To associate a standards file with the current drawing**

1. At the Command prompt, enter **standards**.
2. In the Configure Standards dialog box, Standards tab, click the plus (+) button (Add Standards File).
3. In the Select Standards File dialog box, locate and select a standards file.
4. Click Open.
5. (Optional) Repeat steps 2 and 3 if you want to associate additional standards files with the current drawing.
6. Click OK.

CAD Standards toolbar

**To remove a standards file from the current drawing**

1. At the Command prompt, enter **standards**.
2. In the Configure Standards dialog box, Standards tab, select a standards file in Standards Files Associated with the Current Drawing.
3. Click the X button (Remove Standards File).
4. (Optional) Repeat steps 2 and 3 if you want to remove additional standards files.
5. Click OK.

CAD Standards toolbar

**To change the order of standards files associated with the current drawing**

1. At the Command prompt, enter **standards**.
2. In the Configure Standards dialog box, Standards tab, select the standards file whose position you want to change in Standards Files Associated with the Current Drawing.

3 Do one of the following:

- Click the up arrow button (Move Up) to move the standards file up one position in the list.
- Click the down arrow button (Move Down) to move the standards file down one position in the list.

4 (Optional) Repeat steps 2 and 3 if you want to change the position of other standards files in the list.

5 Click OK.

CAD Standards toolbar

**To specify which standards plug-ins to use when auditing a drawing**

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

2 In the Configure Standards dialog box, Plug-ins tab, do one of the following:

- Select the check box for at least one plug-in to audit a drawing for standards violations.
- To select all plug-ins, right-click in the Plug-ins list. Click Select All. (You can clear all plug-ins. Right-click in the Plug-ins list. Click Clear All.)

3 Click OK.

**CAD Standards toolbar**

# Check Drawings for Standards Violations

You can audit a drawing file for violations of standards and then fix violations. The Batch Standards Checker audits more than one file at a time.

After you associate a standards file with a drawing, you should periodically check the drawing to make sure it conforms with its standards. This is especially important when more than one person is updating the drawing file. For example, in a project with multiple subcontractors, one subcontractor may

create new layers that do not comply with the standards you have defined. In that case, you need to be able to identify the nonstandard layers and fix them.

You can use the notification feature to alert users that a standards violation has occurred while working in a drawing. This feature allows you to fix a standards violation just after it has occurred, making it easier to create and maintain drawings that comply with standards.

### Check a Single Drawing

You can use the CHECKSTANDARDS command to view all standards violations in the current drawing. The Check Standards dialog box reports each nonstandard object, along with any suggested fixes.

You can choose to fix or ignore each reported standards violation. If you ignore a reported violation, it is flagged in the drawing. You can turn off the display of ignored problems so that they are not reported as violations the next time you audit the drawing.

If no fix is provided for the current standards violation, no item is highlighted in the Replace With list, and the Fix button is not available. If you fix a standards violation that is currently displayed in the Check Standards dialog box, the violation is not removed from the dialog box until you click the Fix or the Next button.

Once the entire drawing has been audited, the Checking Complete message is displayed. This message summarizes the standards violations that were found in the drawing. The message also shows the violations that were fixed automatically, the violations that were fixed manually, and the violations that were ignored.

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**NOTE** When two separate violations are displayed for a nonstandard layer (one violation for the nonstandard layer name and one for the nonstandard layer properties), both violations will be fixed at the same time if you choose to fix either one. This results in a discrepancy in the Checking Complete alert summary: there will be fewer standards violations found than were originally reported in the Check Standards dialog box.

---

### Check Multiple Drawings

You can use the Batch Standards Checker to analyze multiple drawings and summarize standards violations in an HTML report. To run a batch standards audit, you must first create a standards check (CHX) file. The CHX file is a configuration and report file; it contains a list of drawing and standards files, as well as a report on a standards check.

By default, each drawing is checked against the standards files that are associated with it. Alternatively, you can override the default and choose another set of standards files to be used.

After the batch standards audit is complete, you can view an HTML report with details of the audit. You can also create notes that are included in the HTML report. This report can be exported and printed. In a collaborative environment, you can distribute the report to drafters so that they can fix any problems with their sections.

### Use Notification of Standards Violations While Working in a Drawing

You can set options for notification in the CAD Standards Settings dialog box and with the STANDARDSVIOLATION system variable. If you select Display Alert upon Standards Violation in the dialog box, then an alert is displayed when a violation occurs while you work. If you select Display Standards Status Bar Icon, an icon is displayed when you open a file associated with a standards file and when you create or modify nonstandard objects.

By default, if an associated standards file is missing or if a violation occurs while you work, a pop-up message appears in the lower-right corner of the application window (the status bar tray).

You should check a drawing for standards violations using the Check Standards dialog box before using the notification options. This prevents triggering a notification alert carried over from a previous session. After a drawing has been checked and fixed, the notification options trigger an alert only if a new violation occurs.

### Display Alerts for Named Objects

If you select Display Alert upon Standards Violation, you are notified of violations when you create or edit only named objects (linetypes, text styles, layers, and dimensions). Standards violations that do not affect named objects do not trigger a notification alert. Also, if a named object was marked as ignored in the Check Standards dialog box, that object no longer triggers a notification alert, even though the named object is nonstandard. Changing nonstandard named objects, such as setting a nonstandard layer as current, does trigger a notification alert.

After an alert is displayed, you can choose to fix or not fix the violation. Choosing to fix the violation opens the Check Standards dialog box. If it is already open, the dialog box addresses the specific violations that just occurred. You can resume your previous work in the Check Standards dialog box after you respond to the most recent standards violations. Alternatively, if you do not want to fix violations when alerted, you can click Do Not Fix to dismiss the alert.

If you open a drawing that has one or more associated standards files, an Associated Standards File(s) icon is displayed in the status bar. If an associated standards file is missing, a Missing Standards File(s) icon is displayed in the status bar. If you double-click the Missing Standards File(s) icon and then resolve or dissociate missing standards files, the Missing Standards File(s) icon is replaced by the Associated Standards File(s) icon.

**NOTE** If you select Display Alert upon Standards Violation in the Check Standards dialog box and you fix a violation, you can return to a fixing operation where you left off. If you select Display Standards Status Bar icon and you click the icon and fix a violation, you must return to a standards-fixing operation from the beginning.

**To audit a drawing for standards violations**

1. Open a drawing that has one or more associated standards files.

   An Associated Standards File(s) icon is displayed in the status bar. If an associated standards file is missing, a Missing Standards File(s) icon is displayed in the status bar.

   **NOTE** If you click the Missing Standards File(s) icon and then resolve or dissociate missing standards files, the Missing Standards File(s) icon is replaced by the Associated Standards File(s) icon.

2. In the drawing with one or more associated standards files, enter **checkstandards** at the Command prompt.

   The Check Standards dialog box is displayed with the first standards violation reported under Problem.

3. Do one of the following:

   - If you want to apply the item selected in the Replace With list to fix the violation reported under Problem, click Fix. If there is a recommended fix in the Replace With list, it is preceded by a check mark. If no recommended fix exists for the current standards violation, the Fix button is not available.

     The Check Standards dialog box automatically displays the next standards violation under Problem.

   - Manually fix the standards violation. Click the Next button to display the next violation.

   - Select Mark This Problem as Ignored. Click the Next button to display the next violation.

     Selecting Mark This Problem as Ignored flags the standards violation so you can prevent it from being displayed the next time you use the CHECKSTANDARDS command.

- Click the Next button to display the next violation.

4. Repeat steps 2 and 3 until you have cycled through all standards violations.
5. Click Close.

CAD Standards toolbar

**To turn the display of ignored problems on or off**

1. In a drawing with one or more associated standards files, enter **checkstandards** at the Command prompt.
2. In the Check Standards dialog box, click Settings.
3. In the CAD Standards Settings dialog box, select or clear Show Ignored Problems.
4. Click OK.

CAD Standards toolbar

**To turn notification for standards violations on or off**

1. At the Command prompt, enter **standards**.
2. In the Configure Standards dialog box, click Settings.
3. Do one of the following:
   - To turn off standards notifications, select Disable Standards Notifications.
   - To be notified by alert, select Display Alert upon Standards Violation.
   - To be notified by the display of an icon in the status bar, select Display Standards Status Bar Icon.
4. Click OK.

CAD Standards toolbar

**To start the Batch Standards Checker**

■ Click Start menu (Windows) ➤ All Programs ➤ Autodesk ➤ AutoCAD 2006 ➤ Batch Standards Checker.

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**NOTE** When you start the Batch Standards Checker from a DOS command line using the DWGCHECKSTANDARDS command, the Batch Standards Checker accepts a variety of command-line parameters that can be used for scripting batch standards audits.

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**To create a standards check file for the Batch Standards Checker**

1. Start the Batch Standards Checker.
2. In the Batch Standards Checker, click File menu ➤ New Check File. (You can also click New on the Batch Standards Checker toolbar.)
3. On the Drawings tab, click the + button (Add Drawing).
4. In the Open dialog box, select a drawing to audit.
5. (Optional) Repeat steps 3 and 4 if you want to add additional drawings to the standards check file.
6. Click File menu ➤ Save As. (You can also click the Save As button on the Batch Standards Checker toolbar.)
7. In the Save As dialog box, in File Name, enter a name for the file.
8. Click Save.

**To open an existing standards check file**

1. Start the Batch Standards Checker.
2. In the Batch Standards Checker, click File menu ➤ Open Check File. (You can also click Open on the Batch Standards Checker toolbar.)
3. In the Open dialog box, locate and select a standards check file.
4. Click Open.

### To specify a standards override for a standards check file

1. Start the Batch Standards Checker.
2. Create a standards check file or open an existing one.
3. In the Batch Standards Checker, Standards tab, select the Check All Drawings Using the Following Standards Files option.
4. Click the + button (Add Standards File).
5. In the Open dialog box, select a standards file to use as an override. Click Open.
6. (Optional) Repeat steps 5 and 6 if you want to add additional standards files as overrides.
7. In the Batch Standards Checker, click File menu ➤ Save Check File. (You can also click Save on the Batch Standards Checker toolbar.)

### To audit a set of drawings for standards violations

1. Start the Batch Standards Checker.
2. Open an existing standards check file, or create a new one.
3. In the Batch Standards Checker, click Check menu ➤ Start Check. You can also click Start Check on the Batch Standards Checker toolbar.)

   When the batch audit is complete, the report is displayed in a browser window.

   You can cancel the batch audit at any time by clicking Stop Check.

   **NOTE** You cannot check encrypted files using the Batch Standards Checker.

### To add notes to the batch audit report

1. Start the Batch Standards Checker.
2. Create a standards check file or open an existing one.
3. In the Batch Standards Checker, Notes tab, enter notes to be included in the report.
4. Click File menu ➤ Save Check File. (You can also click Save on the Batch Standards Checker toolbar.)

### To view a previously generated batch audit report

1. Start the Batch Standards Checker.

2 Open the standards check file that generated the report you want to view.

3 In the Batch Standards Checker, click Check menu ➤ View Report. (You can also click View Report on the Batch Standards Checker toolbar.)

The report is displayed in a browser window.

4 Select one of the following options to filter the data displayed in the report:

- **Overview.** Summarizes the number of problems encountered in each audited drawing.
- **Plug-ins.** Summarizes the standards plug-ins installed on the system used to run the batch audit.
- **Standards.** Summarizes the standards files used to check the drawings in the batch audit.
- **Problems.** Provides detailed information about each problem that was encountered.
- **Ignored Problems.** Provides detailed information about problems that were flagged as ignored.
- **All.** Shows all available standards audit information.

5 In the For list, select individual drawings to view their audit information.

#### To export a batch audit report

1 Start the Batch Standards Checker.

2 Create a standards check file or open an existing one.

3 In the Batch Standards Checker, click Check menu ➤ Export Report. (You can also click Export Report on the Batch Standards Checker toolbar.)

# Translate Layer Names and Properties

With the Layer Translator, you can change a drawing's layers to match layer standards you set.

# Convert Layers to Established Drawing Standards

Using the Layer Translator, you can convert layers in one drawing to standards you define.

For example, if you receive a drawing from a company that does not follow your company's layer conventions, you can convert the drawing's layer names and properties to your company's standards. You can map layers in the drawing you are currently working on to different layers in another drawing or standards file, and then convert the current layers using those mappings. If the drawings contain layers of the same name, the Layer Translator can automatically change the properties of the current layers to match those in the other layers.

You can save your layer translation mappings in a file and reuse them with other drawings.

**To convert a drawing's layers to standard layer settings**

1. Click Tools menu ➤ CAD Standards ➤ Layer Translator.
2. In the Layer Translator, do one of the following:

   - Click Load to load layers from a drawing, a drawing template, or a drawing standards file. In the Select Drawing File dialog box, select the file you want. Click Open.
   - Click New to define a new layer. In the New Layer dialog box, enter a name for the new layer and select its properties. Click OK.

   You can repeat step 2 as many times as you want. If you load other files containing layers of the same name as ones already shown in the Translate To list, the layer properties first loaded in the list are maintained; any duplicate layer properties are ignored.
3. Map layers in the current drawing to the layers you want to convert to. Use either or both of the following methods to map layers:

   - To map all identically named layers from one list to the other, click Map Same.
   - To map individual layers, in the Translate From list, select one or more layers. In the Translate To list, select the layer whose properties you want to use. Click Map to define the mapping. You can repeat this method for each layer or group of layers to be translated.

   To remove a mapping, select the mapping from the Layer Translation Mappings list. Click Remove. To remove all mappings, right-click in the list. Click Remove All.

4 (Optional) You can perform the following tasks in the Layer Translator:

- To change the properties of a mapped layer, in the Layer Translation Mappings list, select the mapping whose properties you want to change. Click Edit. In the Edit Layer dialog box, change the linetype, color, lineweight, or plot style of the mapped layer. Click OK.
- To customize the process of layer translation, click Settings. In the Settings dialog box, select the options you want. Click OK.
- To save layer mappings to a file, click Save. In the Save Layer Mappings dialog box, enter a file name. Click OK.

5 Click Translate to perform the layer translations you have specified.

CAD Standards toolbar

# View Selected Drawing Layers

With the Layer Translator, you can control which layers are visible in the drawing area.

You can choose to display objects on all layers in your drawing, or objects on layers you specify. By viewing selected layers, you can visually verify the contents of those layers.

**To specify which layers are displayed in the drawing area**

1 Click Tools menu ➤ CAD Standards ➤ Layer Translator.

2 In the Layer Translator, click Settings.

3 In the Settings dialog box, do one of the following:

- To display objects on selected layers in your drawing, select Show Layer Contents When Selected. Only the layers selected in the Layer Translator dialog box are displayed in the drawing area.
- To display objects on all layers in your drawing, clear Show Layer Contents When Selected.

4 Click OK.

CAD Standards toolbar

# Purge Unreferenced Layers

You can use the Layer Translator to purge (delete all) unreferenced layers from a drawing.

For example, if your drawing includes layers that you do not need, you may want to remove those layers. Reducing the number of layers makes working with the remaining layers more manageable.

**To purge all unreferenced layers from a drawing**

1. Click Tools menu ➤ CAD Standards ➤ Layer Translator.
2. In the Layer Translator, right-click in the Translate From list. Click Purge Layers. All unreferenced layers are deleted from the current drawing.

CAD Standards toolbar

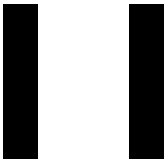
# Part 3
# Control the Drawing Views

Chapter 11 Change a View in Two Dimensions

Chapter 12 Specify a 3D View

Chapter 13 Display Multiple Views in Model Space

# Change a View in Two Dimensions



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.

As you edit your drawing, you can control the drawing display and move quickly to different areas of your drawing while you track the overall effect of your changes. You can zoom to change magnification or pan to reposition the view in the drawing area, and you can save a view and then restore it when you need to plot or refer to specific details.

## In this chapter


- Pan a View
- Magnify a View (Zoom)
- Pan and Zoom with the Aerial View Window
- Save and Restore Views

# Pan a View

You can shift the location of your view by using PAN or by using the window scroll bars.

With the Realtime option of , 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.

By right-clicking, you can display a shortcut menu with additional viewing options.

**See also:**

"Pointing Device Buttons" on page 94

### To pan by dragging

1. Click View menu ➤ Pan ➤ Realtime.
2. When the hand cursor appears, drag the view by holding down the button as you move the pointing device.

   If you are using a wheel mouse, hold down the wheel button and move the mouse.

Standard toolbar

### 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.

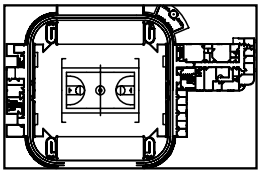
# Magnify a View (Zoom)

When you are working with the details in your drawing, you can zoom in for a closer view.

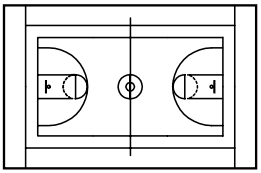
You can change the magnification of a view by zooming in and out, which is similar to zooming in and out with a camera.  does not change the absolute size of objects in the drawing; it changes only the magnification of the view.

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

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



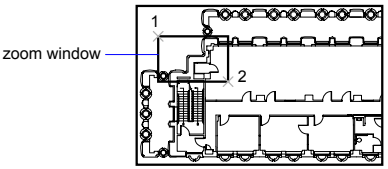
zoomed out



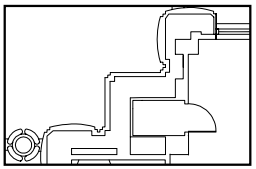
zoomed in

### Zoom to Magnify a Specified Rectangular Area

You can quickly zoom on a rectangular area of your drawing by specifying two diagonal corners of the area you are interested in.




original view



new view

The lower-left corner of the area you specify becomes the lower-left corner of the new display. The shape of the zoom area you specify does not correspond exactly to the new view, which must fit the shape of the viewport.

### Zoom in Real Time

With the Realtime option, you zoom dynamically by moving your pointing device up or down. By right-clicking, you can display a shortcut menu with additional viewing options.

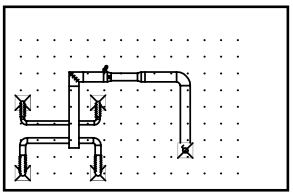
### Zoom to Magnify One or More Objects

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

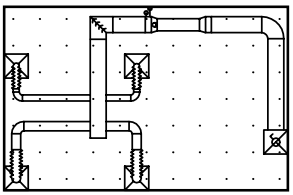
### Zoom to View All Objects in the Drawing

ZOOM Extents displays a view with the largest possible magnification that includes all of the objects in the drawing. This view includes objects on layers that are turned off but does not include objects on frozen layers.

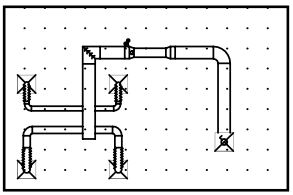
ZOOM All displays either the user-defined grid limits or the drawing extents, whichever view is larger.



current view



zoomed to extents



zoomed to show entire drawing (all)

**See also:**

"Scale Views in Layout Viewports" on page 216
"Toolbars" on page 42

### To zoom by dragging

1. Click View menu ➤ Zoom ➤ Realtime.
2. Hold down the button on your pointing device and drag vertically to zoom in and out.
3. Press ENTER, press ESC , or right-click to exit.

Standard toolbar

**To zoom in to an area by specifying its boundaries**

1. Click View menu ➤ Zoom ➤ 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 entering the ZOOM command.

Standard toolbar

**To display the drawing extents by zooming**

■ Click View menu ➤ Zoom ➤ 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.

Standard toolbar

**To display the area of the grid limits by zooming**

■ Click View menu ➤ Zoom ➤ 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.

Standard toolbar

**To restore the previous view**

■ Click View menu ➤ Zoom ➤ Previous.

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

Standard toolbar

With the ZOOM command active, right-click in the drawing area. Click Zoom Previous.

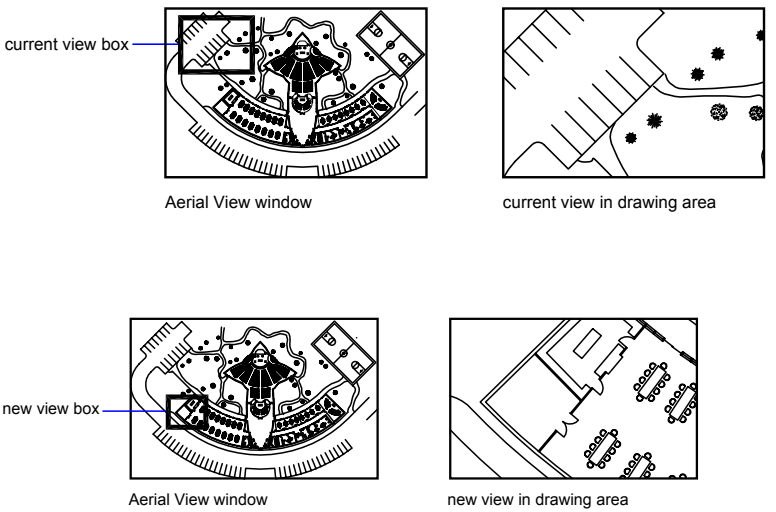
# Pan and Zoom with the Aerial View Window

In a large drawing, you can pan and zoom quickly in a window that can display the whole drawing.

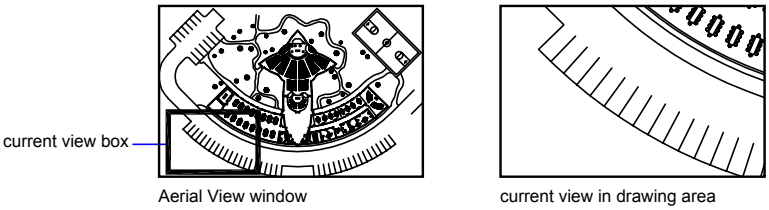
You can use the Aerial View window to change the view in your current viewport quickly. If you keep the Aerial View window open as you work, you can zoom and pan without interrupting your current command. You can also specify a new view without having to choose a menu option or enter a command.

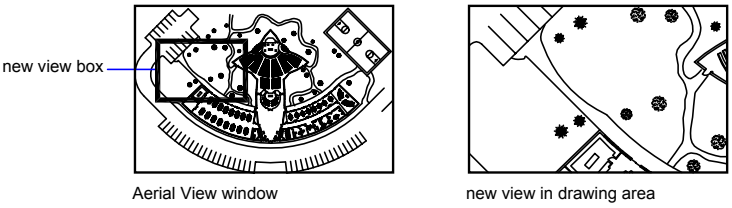
### Use the View Box to Pan and Zoom

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

■ 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

■ 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.

# Save and Restore Views

When you save specific views by name, you can restore them for layout and plotting or when you need to refer to specific details.

A specific magnification, position, and orientation is known as a view. 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 the drawing and can be used anytime. 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
- A view category that you assign to the view (optional)
- The location of the view: the Model tab or a specific layout tab
- Whether the view is associated with a viewport on a sheet
- Layer visibility in the drawing at the time the view is saved
- User coordinate system
- 3D perspective and clipping

### 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.

You can specify a layout view to be displayed when the drawing is opened only if you saved the drawing from that specific layout tab.

### To display the previous view

- On the Standard toolbar, click Zoom Previous.

### To display the previous view during PAN Realtime and ZOOM Realtime

■ Right-click in the drawing area. Click Zoom Original.

### To save and name a view

1. If you have more than one viewport in model space, click inside the viewport that contains the view you want to save. If you are working in a layout, select the viewport.
2. Click View menu ➤ Named Views.
3. In the View dialog box, Named Views tab, click New.
4. In the New View dialog box, enter a name for the view.

   The name can be up to 255 characters long and contain letters, digits, and the special characters dollar sign ($), hyphen (–), and underscore (_).
5. (Optional) Enter a view category.

   If the drawing is part of a sheet set, the view categories for the sheet set are listed. You can add a category or select one from the list.
6. Select one of the following options to define the area of the view:

   ■ **Current Display.** Includes all of the drawing that is currently visible.

   ■ **Define Window.** Saves part of the current display. The dialog box closes while you use the pointing device to specify opposite corners of the view. Click the Define View Window button to redefine the window.
7. Click OK to save the new view and exit each dialog box.

### To restore a saved view

1. If you have more than one viewport in model space, click inside the viewport that contains the view you want to replace. If you are working in a layout, select the viewport.
2. Click View menu ➤ Named Views.
3. In the View dialog box, Named Views tab, select the view that you want to restore.
4. Click Set Current.

   The current view setting is indicated by a small pointer icon beside the name in the list and is also displayed next to Current View.

5 Click OK.

View toolbar

**To delete a named view**

1 Click View menu ➤ Named Views.

2 In the View dialog box, select the view you want to delete.

3 Right-click in the drawing area. Click Delete.

4 Click OK.

View toolbar

**To view the list of saved viewport arrangements**

■ Click View menu ➤ Viewports ➤ Named Viewports.

The list of saved viewports is displayed in the Viewports dialog box on the Named Viewports tab.

Viewports toolbar

# Specify a 3D View

# 12

When working in 3D, you'll often want to display several different views so that you can easily verify the 3D effects of your editing.

One of these viewpoints is often an isometric view to reduce the number of visually overlapping objects. From a selected viewpoint, you can create new objects, edit existing objects, or generate a hidden-line or shaded view.

## In this chapter


- Overview of Specifying a 3D View
- View a Parallel Projection in 3D
- Display a Perspective View
- Specify 3D Views Interactively (3D Orbit)

# Overview of Specifying a 3D View

You can display parallel and perspective views with several commands to facilitate constructing and visualizing 3D models.

When working in 3D, you'll often want to display several different views so that you can easily verify the 3D effects of your editing. One of these viewpoints is often an isometric view to reduce the number of visually overlapping objects. From a selected viewpoint, you can create new objects, edit existing objects, or generate a hidden-line or shaded view.

# View a Parallel Projection in 3D

You can view the parallel projection of a 3D model from any point in model space.

## Overview of Viewing Parallel Projections in 3D

To determine the point or angle in model space, you can

- Choose a preset 3D view from a toolbar.
- Enter a coordinate or angles that represent your viewing location in 3D.
- Change to a view of the *XY* plane of the current UCS, a saved UCS, or the WCS.
- Change the 3D view dynamically with your pointing device.
- Set front and back clipping planes to limit the objects being displayed.

Viewing in 3D is available only in model space. If you are working in paper space, you cannot use 3D viewing commands such as , , or  to define paper space views. The view in paper space is always a plan view.

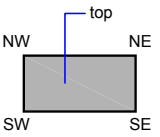
## Choose Preset 3D Views

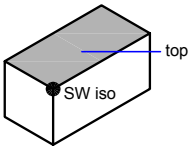
You can select predefined standard orthographic and isometric views by name or description.

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 menu ➤ 3D Views. Select a preset view (Top, Bottom, Left, and so on).

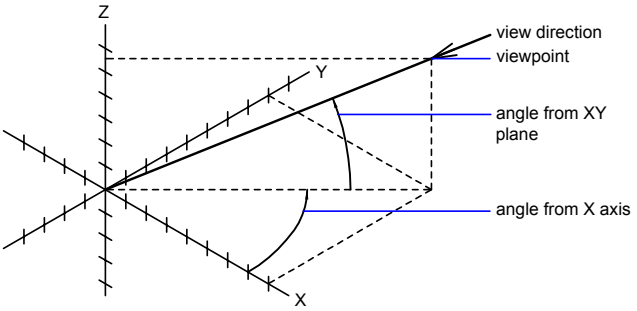
View toolbar

# 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  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 . The following illustration shows a view defined by two angles relative to the *X* axis and the *XY* plane of the WCS.




### To set a view with a viewpoint coordinate

1. Click View menu ➤ 3D Views ➤ Viewpoint.
2. Click inside the compass to specify the viewpoint. The selected viewpoint is used to view the drawing in the direction of 0,0,0.

### To set a view with two angles of rotation

1. At the Command prompt, enter **vpoint**.
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. On the command line, enter **vpoint**.
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. On the command line, enter **vpoint**.
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.

# 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. Click Plan View.
2. Select one of the following options:
   - Current (for the current UCS)
   - World (for the WCS)
   - Named (for a saved UCS)

**NOTE** PLAN changes the viewing direction and turns off perspective and clipping; it does not change the current UCS. Any coordinates entered or displayed subsequent to the PLAN command remain relative to the current UCS.

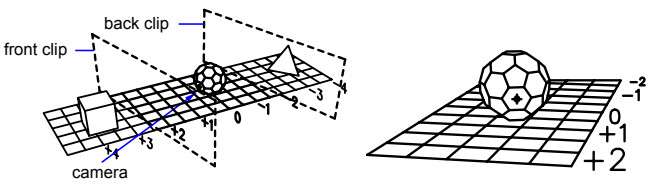
# Change a 3D View Dynamically

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

With dynamic viewing, you can display the effects of changing your viewpoint as you make the changes. Using this method, you can also simplify your view temporarily by choosing only the objects that you need to determine the view. Alternatively, if you press ENTER without selecting any objects, 3D Dynamic View displays a model of a small house instead of your actual drawing. You can use this house to define the viewing angle and distance. When your adjustments are complete and you exit the command, the changes are applied to the entire 3D model in the current view.

### Set Clipping Planes

You can create cutaway, or section, views of your drawing by positioning front and back clipping planes that control the visibility of objects based on their distance from a theoretical camera. You can move the clipping planes perpendicular to the line of sight between the camera and target (where the camera is pointing). Clipping removes the display of objects from the front and back of clipping planes. The following illustration shows how clipping planes work:




### To create a parallel projection dynamically

1. On the command line, enter **dview**.
2. Do *one* of the following:
   - Select the objects to display and press ENTER.
   - Press ENTER without selecting objects to see a representation of a house that shows the current viewing angle.
3. Enter **ca** (Camera).

By default, a camera point is set at the center of the drawing.

4 Adjust the view as if you're aiming a camera.

You can set your view dynamically by moving the crosshairs and clicking.

5 To switch between the angle-input methods, enter **t** (Toggle Angle).

You can also adjust the view with one of two angle-input methods.

- For the Enter Angle from the XY Plane option, enter the angle of the camera up or down relative to the *XY* plane of the current UCS. The default setting, 90 degrees, points the camera straight down from above.

  After you enter the angle, the camera is locked at that height, and you can rotate the camera about the target with the rotation angle measured relative to the *X* axis of the current UCS.

- For the Enter Angle in the XY Plane from the X Axis option, rotate the camera about the target with the rotation angle measured relative to the *X* axis of the current UCS.

6 When you finish specifying the parallel projection, press ENTER.

### To set clipping planes

1 On the command line, enter **dview**.

2 Select the objects on which to base the view.

3 On the command line, enter **cL** (Clip).

4 Enter **f** to set a front clipping plane or **b** to set a back clipping plane, or press ENTER.

5 Position the clipping plane by dragging the slider or entering a distance from the target.

6 Press ENTER to exit the command.

### To restore the default view and UCS

1 Click View menu ➤ 3D Views.

2 Click Plan View.

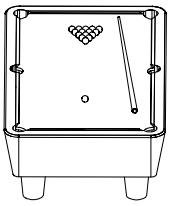
3 Click World UCS.

# Display a Perspective View

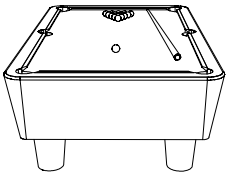
You can define perspective views of a model to create realistic effects.

The difference between defining perspective views and defining parallel projections is that perspective views require a distance between a theoretical camera and target point. Small distances produce severe perspective effects; large distances produce mild effects.

The following illustration shows the same wireframe model in a parallel projection and in a perspective view. Both are based on the same viewing direction.



parallel projection



perspective projection

Many operations are not available in perspective view, including panning and zooming and operations that require using object snap or input from the pointing device.

A perspective view remains in effect until the perspective effect is turned off or until a new view is defined in its place.

**To define a perspective view of a 3D model**

1. On the command line, enter **dview**.
2. Select the objects to display.
3. Enter **ca** (Camera).

   By default, a camera point is set at the center of the drawing.
4. Adjust the view as if you're aiming a camera.

   A representation of a house shows the current viewing angle. You can set your view dynamically by moving the crosshairs and clicking.
5. To switch between angle-input methods, enter **t** (Toggle Angle).

You also can adjust the view with one of two angle-input methods.

- For the Enter Angle from the XY Plane option, enter the angle of the camera up or down relative to the *XY* plane of the current UCS. The default setting, 90 degrees, points the camera straight down from above.

  After you enter the angle, the camera is locked at that height, and you can rotate the camera about the target with the rotation angle measured relative to the *X* axis of the current UCS.
- For the Enter Angle in the XY Plane from the X Axis option, rotate the camera about the target with the rotation angle measured relative to the *X* axis of the current UCS.

6 To turn on the perspective view, enter **d** (Distance).

7 Specify a distance, or press ENTER to set the perspective view.

You can use the slider bar to set the distance between the selected objects and the camera, or you can enter a real number. If the target and camera points are very close (or if the Zoom option is set high), you might see only a small part of your drawing.

**To turn off a perspective view**

1 On the command line, enter **dview**.

2 Select the objects to display.

3 Enter **o** (Off).

Perspective is turned off and the view is restored to a parallel projection.

# Specify 3D Views Interactively (3D Orbit)

Use the  command to activate an interactive view of your 3D model.

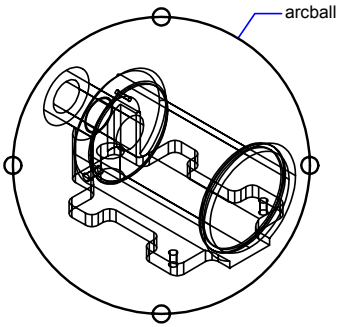
## Overview of 3D Orbit

The  command activates an interactive 3D Orbit view in the current viewport. You use the pointing device to manipulate a view of the entire model or any object in the model from different points around it.

**NOTE** Viewing the entire drawing may degrade the video display.

The 3D Orbit view displays an arcball, which is a circle divided into four quadrants by smaller circles. When 3DORBIT is active, the target of the view

stays stationary and the camera location, or point of view, moves around the target. The center of the arcball, not the center of the objects you're viewing, is the target point.




You can also shade the objects in your drawing while the 3DORBIT command is active.

# Set 3D Graphics Display Options (3D Orbit)

Graphics configuration settings affect the way that 3D objects are displayed; for example, the shading of 3D objects and the way graphics are displayed when the  command is active. You set these options using the 3D Graphics System Configuration dialog box. These settings do not affect how objects are rendered.

This program uses the Heidi® 3D Graphics System developed by Autodesk as the default graphics system. If you want to use a different graphics system, install it according to your display card's vendor documentation.

# Set Lights and Materials (3D Orbit)

You can adjust how lights, materials, textures, and transparency are displayed in 3D views by changing the settings in the "3D Graphics System Configuration Dialog Box". These settings affect objects in the 3D Orbit view and objects shaded using . You have several options:

- **Enable Lights .** Controls the illumination, as defined by .
- **Enable Materials .** Controls the materials attached using .

- **Enable Textures .** Controls textured materials attached using RMAT and .
- **Transparency .** Controls image quality at the expense of drawing speed.

The display both in 3D Orbit and with objects that are shaded using SHADEMODE is subject to the following limitations.

For lights:

- Shadows are not shown.

For materials:

- 2D textures (bitmaps and bitmap blending) are displayed for the color/pattern material component only.
- 3D textures (also called solid, template, or procedural materials) are not shown.
- Bump maps are not shown.
- Opacity maps are not shown.
- Refraction is not shown.
- Reflection is not shown.

Other rendering features, such as background and fog, are not supported in 3D views.

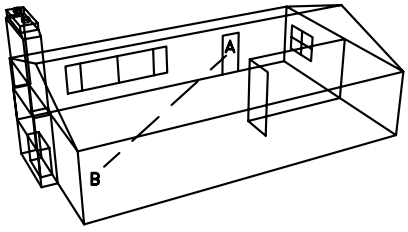
**See also:**

**"Use Lights in Rendering"**

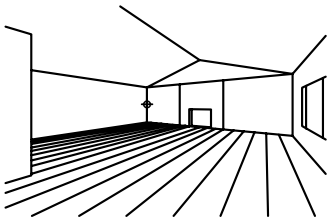
# Locate the Camera Position (3D Orbit)

You use the  command to change the point from which you view the objects in a 3D view and the point that you are viewing. The point from which you are viewing objects is the camera location, and the point you are viewing is the target location.

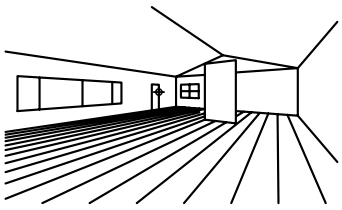
When the  command is active, the default target location is the center of the 3D view, which is not always the center of the objects you are viewing. Use CAMERA before starting 3DORBIT to change the camera and target locations for the 3D Orbit view.




camera location and target point



camera at A, target at B



camera at B, target at A

**To change the camera and target locations**

1. On the command line, enter **camera**.

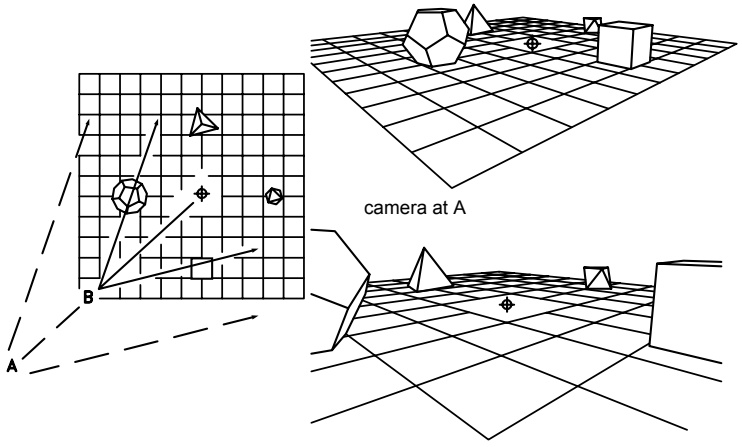
   The current camera and target positions are displayed.
2. To set the camera location, use the pointing device or enter *X*,*Y*,*Z* coordinate values.
3. To set the target location, use the pointing device or enter *X*,*Y*,*Z* coordinate values.

**To adjust the camera distance in the 3D Orbit view**

1. Click View menu, ➤ 3D Orbit.
2. Right-click in the drawing area. Click More ➤ Adjust Distance.

   The cursor changes to a line with one arrow pointing up and one pointing down.
3. Click and drag toward the top of the screen to move the camera closer to the objects. Click and drag toward the bottom of the screen to move the camera away from the objects.

   Adjust Distance simulates the effect of moving the camera closer to the objects or farther away.




# Use 3D Orbit Commands

You can choose from several commands that control the 3D Orbit display, projection, and visualization tools. You can access Pan and Zoom options from the 3D Orbit shortcut menu or on the 3D Orbit toolbar. You can also choose to use a perspective or a parallel projection of the view while  is active.

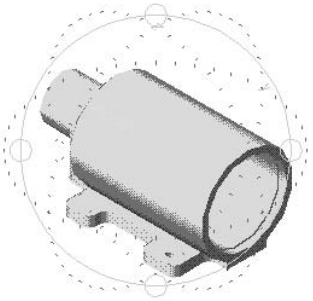
### Shade Objects in the 3D Orbit View

Objects in the 3D Orbit view can be shaded to give them a more realistic 3D appearance. You can change the way objects are shaded using the different shading modes, which are also accessible from the 3D Orbit shortcut menu.

### Use Visual Aids in the 3D Orbit View

You can choose to display one or more of the visual aids (compass, grid, and UCS icon) in the 3D Orbit view. A check mark is displayed next to the active visual aid option(s). A visual aid that is active when you exit 3DORBIT remains active in the view outside the 3D Orbit view unless  is set to 2D Wireframe.

- **Compass**. Draws a sphere within the arcball composed of three lines representing the *X*, *Y*, and *Z* axes.



- **Grid.** Draws an array of lines on a plane parallel to the current *X* and *Y* axes, perpendicular to the *Z* axis. You specify the height of the grid display in the  system variable.

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

  When you zoom in and out of the 3D Orbit view, the number of grid lines changes to give you a clear view of the lines. Fewer lines are drawn as you zoom out. As you zoom in closer, more grid lines are drawn until the number of lines corresponds again to the number set in GRIDUNIT.

- **UCS Icon**. Turns the display of the UCS icon on and off. If the UCS icon is displayed when you start 3DORBIT, a shaded 3D UCS icon is displayed in the 3D Orbit view. On the 3D UCS icon, the *X* axis is red, the *Y* axis is green, and the *Z* axis is blue or cyan. The  command also controls the display of the 3D UCS icon.

#### Use Continuous Orbit

You can click and drag in the 3D Orbit view to start a continuous motion. When you release the pick button on your pointing device, the orbit continues in the direction that you were dragging.

While Continuous Orbit is active, you can change the view by right-clicking in the drawing area and choosing Projection, Shading Modes, Visual Aids, Reset View, or Preset Views from the shortcut menu. You can also turn the front and back clipping planes on and off while Continuous Orbit is active; however, you cannot adjust the clipping planes. If you choose Pan, Zoom, Orbit, or Adjust Clipping Planes from the shortcut menu, Continuous Orbit ends.

### Reset the View or Use Preset Views

While 3DORBIT is active, you can reset the view to the view that was current when you first entered the 3D Orbit view.

You can set the view to one of the six standard orthogonal views, or one of four isometric views. The isometric views are based on a camera location of 0,0,0.

### Access 3D Orbit Commands from the Command Line

You cannot enter commands on the command line while 3DORBIT is active. If 3DORBIT is not active, you can enter a command that starts 3DORBIT and activates one of the options at the same time. For example,  starts 3D Orbit view and activates the Zoom option. See , 3DORBIT, , , , and 3DZOOM in the *Command Reference*.

### To start 3D Orbit view

1. Select the object or objects you want to view with 3DORBIT.

   You can view your entire model by not selecting any objects. However, viewing only selected objects enhances performance.

   **NOTE** OLE objects and raster objects do not appear in the 3D Orbit view.

2. Click View menu ➤ 3D Orbit.

   An arcball is displayed within the active viewport. If the UCS icon is on, it appears shaded. If the GRID is on, a 3D line grid replaces the 2D dot grid.

3. Click and drag the cursor to rotate the view. When you move your cursor over different parts of the arcball, the cursor icon changes. The appearance of your cursor when you click to start dragging indicates the rotation of the view as follows:

   - A small sphere encircled by two lines is the cursor icon displayed when you move the cursor inside the arcball. By clicking and dragging when the cursor is a sphere you can manipulate the view freely. It works as if your cursor were grabbing a sphere surrounding the objects and dragging the sphere around the target point. You can drag horizontally, vertically, and diagonally.

- A circular arrow around a small sphere is the cursor icon displayed when you move the cursor outside the arcball. Clicking outside the arcball and dragging the cursor around the arcball moves the view around an axis that extends through the center of the arcball, perpendicular to the screen. This is called a "roll."

  If you drag the cursor into the arcball, it changes to a small sphere encircled by two lines and the view moves freely, as described above. If you move the cursor back outside the arcball, you revert to a roll.

- A horizontal ellipse around a small sphere is the cursor icon displayed when you move the cursor over one of the smaller circles on the left or right side of the arcball. Clicking and dragging from either of these points rotates the view around the vertical or *Y* axis that extends through the center of the arcball. The *Y* axis is represented on the cursor by a vertical line.

- A vertical ellipse around a small sphere is the cursor displayed when you move the cursor over one of the smaller circles on the top or bottom of the arcball. Clicking and dragging from either of these points rotates the view around the horizontal or *X* axis that extends through the center of the arcball. The *X* axis is represented on the cursor by a horizontal line.

**NOTE** You cannot edit objects while 3DORBIT is active. To exit 3DORBIT, press ENTER or ESC , or click Exit on the shortcut menu.

**To pan in the 3D Orbit view**

1. Click View menu ➤3D Orbit.
2. Right-click in the drawing area. Click Pan.

   The cursor changes to a hand cursor.

3 Click and drag the cursor to move the view horizontally, vertically, or diagonally.

The view moves in the direction that you drag the cursor.

You can use the projection options, shading modes, and visual aids at the same time that you pan and zoom by choosing an option from the 3D Orbit shortcut menu.

4 To stop panning, right-click in the drawing area. Click Orbit or Zoom.

### To zoom in the 3D Orbit view

1 Click View menu ➤ 3D Orbit.

2 Right-click in the drawing area. Click Zoom.

The cursor changes to a magnifying glass with plus (+) and minus (–) signs. Zoom simulates the effect of a camera's zoom lens by making objects appear closer or farther away. Zooming in magnifies the image. If you are using perspective projection, Zoom exaggerates the perspective with which you view the objects. It may slightly distort the display of some objects.

3 Click and drag toward the top of the screen to zoom in. Click and drag toward the bottom of the screen to zoom out.

4 To change the projection or shading mode or use a visual aid, right-click and click an option on the shortcut menu.

5 To stop zooming, right-click in the drawing area. Click Orbit or Pan.

### To use a zoom window in the 3D Orbit view

1 Click View menu ➤ 3D Orbit.

2 Right-click in the drawing area. Click More ➤ Zoom Window.

The cursor changes to a window icon, and you can select a specific area that you want to zoom.

3 Click and drag the pointing device to draw a square around the area you want to select.

When you release the pick button, the drawing is zoomed in, focused on the area you selected.

### To zoom to extents in the 3D Orbit view

1 Click View menu ➤ 3D Orbit,

2 Right-click in the drawing area. Click More ➤ Zoom Extents.

The view is centered and sized to display all objects in the 3D view.

### To choose a parallel or perspective projection in the 3D Orbit view

1 Click View menu ➤ 3D Orbit.

2 Right-click in the drawing area. Click Projection.

3 Select one of the following options:

- **Parallel .** Changes the view so that two parallel lines never converge at a single point. The shapes in the drawing always remain the same and do not appear distorted when they are closer. This is the default.
- **Perspective .** Changes the view so that all parallel lines converge at one point. Objects appear to recede into the distance while parts of the objects appear larger and closer to you. The shapes are somewhat distorted when the object is very close. This view correlates most closely to what your eye sees.

A check mark is displayed next to the current projection option.

**NOTE** When you exit 3DORBIT, the projection that you choose remains current. You cannot edit, pick points, zoom, or pan when viewing objects in perspective projection.

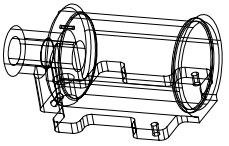
### To change the shading mode in 3D Orbit view

1 Click View menu ➤ 3D Orbit.

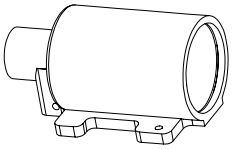
2 Right-click in the drawing area. Click Shading Modes.

3 Select one of the following options:

- **Wireframe .** Displays the objects in the 3D view using lines and curves to represent the boundaries.



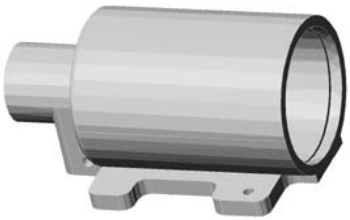
- **Hidden**. Displays the objects in the 3D view using wireframe representation with the lines representing the back faces hidden.



- **Flat Shaded .** Shades the objects in the 3D view between the polygon faces. This gives the objects a faceted, less smooth appearance.

- **Gouraud Shaded .** Shades the objects in the 3D view and smooths the edges between polygon faces. This gives the objects a smoother, more realistic appearance.

- **Flat Shaded, Edges On .** Combines the Flat Shaded and Wireframe options. The objects are flat shaded with the wireframe showing through.



- **Gouraud Shaded, Edges On .** Combines the Gouraud Shaded and Wireframe options. The objects are Gouraud shaded with the wireframe showing through.

**NOTE** When you shade objects in the 3D Orbit view, the shading is still applied to the objects after you exit 3DORBIT. Use SHADEMODE to change the shading when 3DORBIT is not active.

### To display a visual aid

1. Click View menu ➤ 3D Orbit.
2. Right-click in the drawing area. Click Visual Aids.
3. Select one or more of the following options: Compass, Grid, or UCS Icon.

### To start a continuous orbit

1. Click View menu ➤ 3D Orbit.
2. Right-click in the drawing area. Click More ➤ Continuous Orbit.

   The cursor changes to a small sphere encircled by two lines.
3. Click and drag in the direction that you want the continuous orbit to move.
4. Release the pick button.

   The orbit continues moving in the direction that you indicated with your pointing device.
5. To change the direction of the continuous orbit, click and drag in a new direction and release the pick button.

   The direction of the continuous orbit changes.

### To reset the view

1. Click View menu ➤ 3D Orbit.
2. Right-click in the drawing area. Click Reset View.

   The view is reset to the view that was current when you started 3DORBIT.

### To use a preset 3D Orbit view

1. Click View menu ➤ 3D Orbit.
2. Right-click in the drawing area. Click Preset Views.
3. Select one of the views from the list.

# Adjust Clipping Planes (3D Orbit)

You can set the front and back clipping planes for the objects in 3D Orbit view. Anything outside the clipping planes is not visible.

You can set clipping planes for the objects in 3D Orbit view. A *clipping plane* is an invisible plane. Objects or parts of objects that move beyond a clipping plane cannot be seen in the view. In the Adjust Clipping Planes window, there are two clipping planes, front and back. The front and back clipping planes are represented as lines at the top and bottom of the Adjust Clipping Planes window.

You use the toolbar buttons or the options on the Adjust Clipping Planes shortcut menu to choose the clipping plane that you want to adjust.

If clipping planes are on when you exit the 3D Orbit view, they remain on in the 2D and 3D view.

You can adjust only one clipping plane at a time except when using the Create Slice option. On the toolbar, the depressed button indicates the plane that you are adjusting. When you've finished adjusting one plane, you can adjust the other. If the Create Slice button is depressed, you are adjusting both planes at the same time.

### Turn Clipping Planes On and Off

Closing the Adjust Clipping Planes window does not turn the clipping planes off. You can rotate the 3D Orbit view and still see the clipping planes. As you rotate the view, different portions of the objects are clipped as they pass through the clipping planes. You can use the 3D Orbit shortcut menu to turn the clipping planes off and on.

**NOTE** If the clipping planes are on when you exit 3D Orbit view, they remain on in the 2D and 3D view.

**To adjust clipping planes in the 3D Orbit view**

1. Click View menu ➤ 3D Orbit.
2. Right-click in the drawing area. Click More ➤ Adjust Clipping Planes.

   The Adjust Clipping Planes window displays the objects in the view, rotated at a 90-degree angle. You set the clipping planes in the Adjust Clipping Planes window, and the results are displayed in the main 3D Orbit view.
3. Right-click in the Adjust Clipping Planes window. Click one of the following:

   - **Front Clipping On.** Toggles the front clipping plane on or off. When the front clipping plane is on, you can see the results of moving the line that adjusts the front clipping plane, as described in step 4.
   - **Back Clipping On**. Toggles the back clipping plane on or off. When the back clipping plane is on, you can see the results of moving the line that adjusts the back clipping plane, as described in step 4.
4. Right-click in the Adjust Clipping Planes window. Click one of the following options to adjust the clipping planes:

   - **Adjust Front Clipping**. Adjusts only the front clipping plane. The line near the bottom of the window adjusts the front clipping plane. If you chose Front Clipping On in step 3, you can see the clipping in the main 3D Orbit view as you move the line up or down.
   - **Adjust Back Clipping**. Adjusts only the back clipping plane. The line near the top of the window adjusts the back clipping plane. If you chose Back Clipping On in step 3, you can see the clipping in the main 3D Orbit view as you move the line up or down.
   - **Create Slice**. Causes the back and front clipping planes to move together at a set distance apart. Adjust the front and back clipping planes as described above to set the distance, and then choose Create Slice. This displays a "slice" of the objects in the 3D Orbit view. To see the slice in the main 3D Orbit view, choose both Front Clipping On and Back Clipping On in step 3.
5. Click and drag the Clipping Plane line to clip the area that you do not want to see in the view.

   You can adjust only one clipping plane at a time except when using the Create Slice option. On the toolbar, the depressed button indicates the

plane that you are adjusting. When you've finished adjusting one plane, you can adjust the other. If the Create Slice button is depressed, you are adjusting both planes at the same time.

### To turn clipping planes on and off

1. Click View menu ➤ 3D Orbit.
2. Right-click in the drawing area. Click More.

   A check mark in front of Front Clipping On or Back Clipping On indicates that the clipping plane is on.
3. Select the option that has a check mark next to it to turn the clipping plane off.

# Display Multiple Views in Model Space

# 13

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.

You can save arrangements of model space viewports for reuse at any time.

## In this chapter

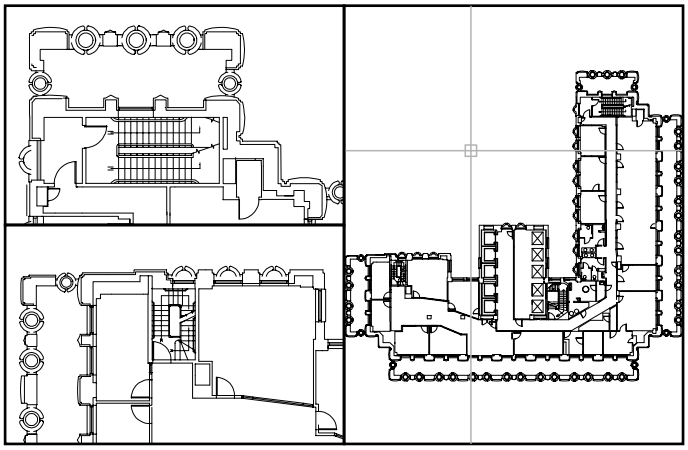

- Set Model Space Viewports
- Select and Use the Current Viewport
- Save and Restore Model Tab Viewport Arrangements

# 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 203.

### Use Model Space Viewports

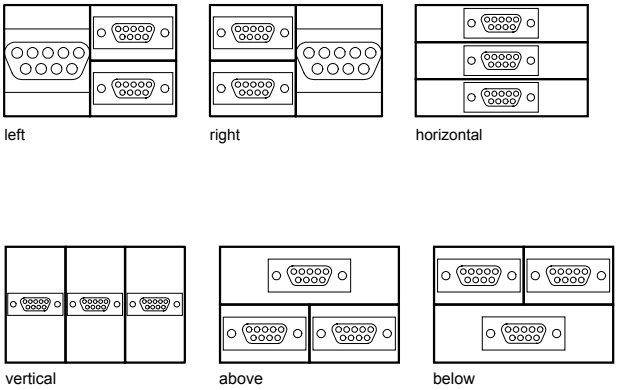
With model space viewports, you can do the following:

- Pan; zoom; set Snap, Grid, and UCS icon modes; and restore named views.
- Save user coordinate system orientations with individual viewports.
- Draw from one viewport to another when executing a command.
- Name a viewport arrangement so that you can reuse it on the Model tab or insert it on a layout tab.

Setting up different coordinate systems in individual viewports is useful if you typically work on 3D models. See "Assign User Coordinate System Orientations to Viewports" on page 331.

### 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. Click View menu ➤ Viewports and then either 2 Viewports, 3 Viewports, or 4 Viewports to indicate how many model space viewports should be created.
3. At the Next prompt, specify the arrangement of the new viewports.

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

1. Click View menu ➤ 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 menu ➤ Viewports ➤ 1 Viewport.

**To switch from a layout tab to the Model tab**

- Click the Model tab at the bottom of the drawing area.

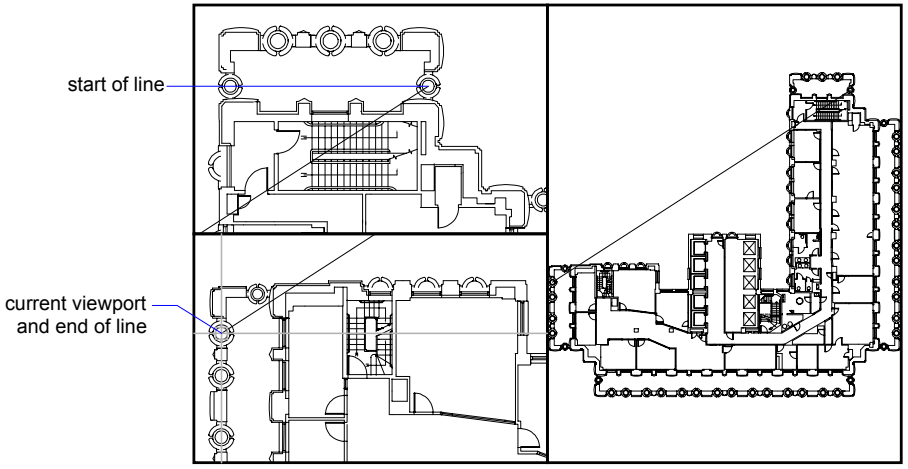
# Select and Use the Current Viewport

When you use multiple viewports, one of them is the *current viewport*, which accepts cursor input and view commands.

When a viewport is current, the cursor is displayed as crosshairs rather than an arrow, and the viewport boundary is highlighted. You can change the current viewport at any time except when a View command is in progress.

To make a viewport the current viewport, you click inside it or press CTRL + R to cycle through the existing viewports.

To draw a line using two model space viewports, you start the line in the current viewport, make another viewport current by clicking within it, and then specify the endpoint of the line in the second viewport. In a large drawing, you can use this method to draw a line from a detail in one corner to a detail in a distant corner.




**To make a viewport current**

- Click anywhere within the viewport border.

**To cycle through viewports without clicking**

- Press CTRL + R repeatedly.

# 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 , 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 menu ➤ Viewports ➤ New Viewports.
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 menu ➤ Viewports ➤ Named Viewports.
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 menu ➤ Viewports ➤ Named Viewports.
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 menu ➤ Viewports ➤ Named Viewports to display the Viewports dialog box.

All saved viewport arrangements in the drawing are listed on the Named Viewports tab under Named Viewports.

# Part 4
# Choose a Work Process Before You Begin

Chapter 14 Create Single-View Drawings (Model Space)

Chapter 15 Create Multiple-View Drawing Layouts (Paper Space)

Chapter 16 Work with Sheets in a Sheet Set

# Create Single-View Drawings (Model Space)

# 14

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

With this method, you create the building, mechanical part, or geographic area that you want to represent at full scale (1:1), but you create the text, dimensions, and the title block of the drawing at a scale to match the intended plot scale.

## In this chapter


- Quick Start for Model Space Drafting
- Draw, Scale, and Annotate in Model Space

# Quick Start for Model Space Drafting

The process of creating and plotting a drawing file in model space is very different from the process used in manual drafting.

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

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

- It is suitable for 2D drawings only
- It does not support multiple views and view-dependent layer settings
- Scaling the annotation and title block requires computation

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.

**See also:**

"Create Multiple-View Drawing Layouts (Paper Space)" on page 203
"Work with Sheets in a Sheet Set" on page 233

# Draw, Scale, and Annotate in Model Space

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

### 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  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 ().
- **Dimensions.** Set the dimension scale in a dimension style () or with the system variable.
- **Linetypes.** Set the scale for noncontinuous linetypes with the  and  system variables.
- **Hatch patterns.** Set the scale for hatch patterns in the Hatch and Gradient dialog box () or with the  system variable.
- **Blocks.** Specify the insertion scale for blocks either as you insert them, or set an insertion scale in the Insert dialog box () or in DesignCenter (). The system variables used for inserting blocks are , , and . This also applies to the border and title block of the drawing.

### Determine the Scale Factor for Plotting

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

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

1/4" = 12"

1 = 12 x 4

1 (plotted unit) = 48 (drawing units)

Using the same calculation, the scale factor for 1 centimeter = 1 meter is 100, and the scale factor for 1 inch = 20 feet is 240.

### Sample Scale Ratios

The sample architectural scale ratios in the table can be used to calculate text sizes in model space.

| Scale | Scale factor | To plot text size at | Set drawing text size to |
|---|---|---|---|
| 1 cm = 1 m | 100 | 3 mm | 30 cm |
| 1/8" = 1'-0" | 96 | 1/8" | 12" |
| 3/16" = 1'-0" | 64 | 1/8" | 8" |
| 1/4" = 1'-0" | 48 | 1/8" | 6" |
| 3/8" = 1'-0" | 32 | 1/8" | 4" |
| 1/2" = 1'-0" | 24 | 1/8" | 3" |
| 3/4" = 1'-0" | 16 | 1/8" | 2" |
| 1" = 1'-0" | 12 | 1/8" | 1.5" |
| 1 1/2" = 1'-0" | 8 | 1/8" | 1.0" |

If you are working in metric units, you might have a sheet size of 210 x 297 mm (A4 size) and a scale factor of 20. You calculate grid limits as follows:

210 x 20 = 4200 mm

297 x 20 = 5900 mm

**See also:**

"Specify Units and Unit Formats" on page 107

### 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.

# Create Multiple-View Drawing Layouts (Paper Space)

# 15

You *layout viewports* to display views of your model. Each view can have a different viewing angle, view scale, and layer display.

## In this chapter

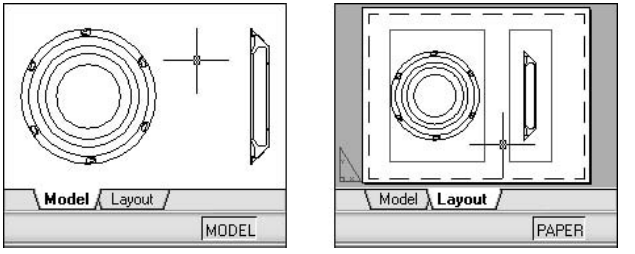

- Quick Start for Layouts
- Understand the Layout Process
- Work with Model Space and Paper Space
- Create and Modify Layout Viewports
- Control Views in Layout Viewports
- Reuse Layouts and Layout Settings

# Quick Start for Layouts

There are two distinct working environments, or "spaces," in which you can create objects in a drawing.

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

Working on the Model tab, you draw a model of your subject at 1:1 scale. Working on a layout tab, you place one or more viewports, dimensions, notes, and a title block to represent a drawing sheet.




On a layout tab, each layout viewport is like a picture frame containing a "photograph" of the model. 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.

# 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.

### 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.

The other topics in this chapter provide additional detail on how to create, use, and modify layouts and layout viewports.

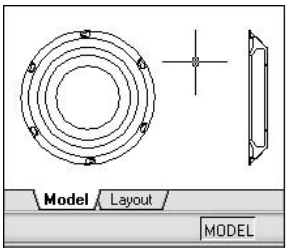
# Work with Model Space and Paper Space

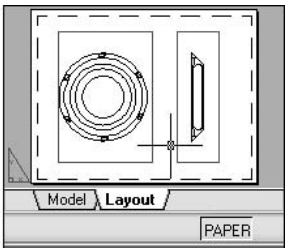
There are several benefits to switching between model space and paper space to perform certain tasks. Use model space for creating and editing your model. Use paper space for composing your drawing sheet and defining views.

## Work on the Model Tab

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:

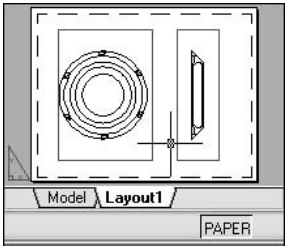
- Click the Model tab.
- Right-click any layout tab or the Model tab. Click Activate Model Tab.

# 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. The crosshairs cursor is active over the entire layout area.



### 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.

Right-click a layout tab to display a layout shortcut menu with options.

---

**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 by selecting the layout and clicking Page Setup on the File menu.

### To specify the page setup for the current layout

■ Right-click the current layout tab. Click Page Setup.

Layouts toolbar

### To create a new layout

1. Click Insert menu ➤ Layout ➤ New Layout.
2. Enter the name of the new layout on the command line.

   A new layout tab is created. To switch to the new layout, choose the layout tab.

Layouts toolbar

### To import a layout from a template

1. Click Insert menu ➤ Layout ➤ Layout from Template.
2. In the Select File dialog box, select a DWT or DWG file to import a layout from.
3. Click Open.
4. In the Insert Layout(s) dialog box, select a layout to import.

   A new layout tab is created. To switch to the new layout, click the layout tab.

Layouts toolbar

### To create a layout using a wizard

1. Click Insert menu ➤ Layout ➤ Layout Wizard.
2. On each page of the Create Layout wizard, select the appropriate settings for the new layout.

When finished, the new layout will be the current layout tab.

**To duplicate a layout**

1. Right-click the layout tab of the layout you want to duplicate. Click Move or Copy.
2. In the Move or Copy dialog box, select a position for the new layout tab.
3. Make sure that Create a Copy is selected.
4. Click OK.

---

**NOTE** The Model tab cannot be duplicated.

---

**To rename a layout**

1. Right-click the layout tab for the layout you want to rename. Click Rename.
2. In the Rename Layout dialog box, enter the new name for the layout.
3. Click OK.

---

**NOTE** The Model tab cannot be renamed.

---

**To delete a layout**

1. Right-click the layout tab for the layout you want to delete. Click Delete.
2. In the warning box, click OK to delete the layout.

   Any named views associated with the layout are deleted automatically when the layout is deleted.

---

**NOTE** The Model tab cannot be deleted.

---

**To rearrange layout tabs**

1. Right-click the layout tab of the layout you want to reposition. Click Move or Copy.
2. In the Move or Copy dialog box, select the layout tab that should be placed immediately after the moved or copied layout tab. If you want to move the layout tab to the end of the list of layout tabs, select Move to End.

3 Click OK.

**NOTE** The Model tab cannot be repositioned.

### To make a layout current

Do one of the following to make a layout current:

- Click the layout tab for the layout that you want to make current.
- Press CTRL+Page Down to cycle through the layout tabs from left to right or CTRL+Page Up to cycle through the layout tabs from right to left. Stop when you get to the tab of the layout that you want to make current.

### To 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.

### To plot a layout

1. Do *one* of the following:
   - Right-click a layout tab. Click Plot.
   - Hold down SHIFT to select several layout tabs. Right-click a layout tab. Click Publish Selected Layouts.
2. Select settings in the Plot or the Publish dialog box. Click OK or Publish.

Standard toolbar

# 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.

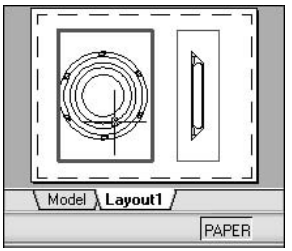
---

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  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  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.

Status bar

,

Right-click. Click Maximize Viewport.

# Create and Modify Layout Viewports

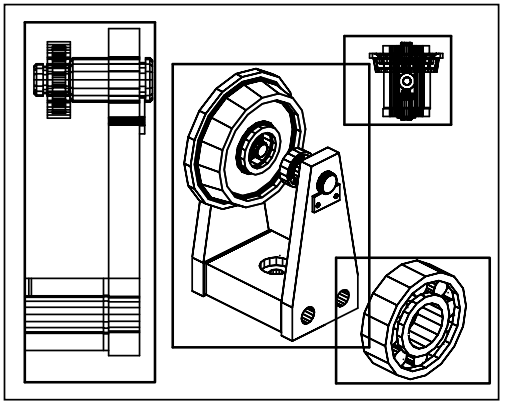
You can create a single layout viewport that fits the entire layout or create multiple layout viewports in the layout. Once you create the viewports, you can change their size, their properties, and also scale and move them as needed.

---

**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 , you have several options for creating one or more layout viewports. You can also use  and  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.

The  command provides two options to assist you in defining an irregularly shaped viewport.

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.

The Polygonal option can be used to create a nonrectangular layout viewport by specifying points. The prompt sequence is the same as the prompt sequence 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 boundary is not displayed, but the viewport is not clipped.

---

#### 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.

#### Clip Layout Viewports

You can redefine the boundary of a layout viewport by using the  command. To clip a layout viewport, you can use your pointing device to either select an existing object to designate as the new boundary, or specify the points of a new boundary.

#### To create a new layout viewport

1. On the layout tab, click View menu ➤ Viewports ➤ 1 Viewport.
2. Click to specify one corner of the new layout viewport.
3. 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.

Viewports toolbar

,

#### To create a viewport configuration on a layout

1. Click a layout tab.

2 Click View menu ➤ Viewports ➤ New Viewports.

3 In the Viewports dialog box, New Viewports tab, 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.

Viewports toolbar

**To place a named viewport configuration into a layout**

1 Click a layout tab.

2 Click View menu ➤ Viewports ➤ New Viewports.

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.

Viewports toolbar

**To modify viewport properties using the Properties palette**

1. Click the border of the layout viewport whose properties you want to modify.
2. Right-click, and then click 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 current viewport.

Standard toolbar

Select the viewport and right-click in the drawing area. Click Properties.

**To clip a viewport boundary**

1. At the Command prompt, enter **vpclip**.
2. Select the viewport to clip.
3. Enter **d** (Delete) to delete the clipping boundary.
4. Enter **p** (Polygonal).
5. Specify points, or select the object, to define the new viewport boundary.

Select the viewport to clip and right-click in the drawing area. Click Viewport Clip.

# 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  command
- The Viewports toolbar

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

---

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

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

#### Lock the Scale of Layout Viewports

Once you set the viewport scale, you cannot zoom within a viewport without changing the viewport scale. By locking the viewport scale first, you can zoom in to view different levels of detail in your viewport without altering the viewport scale.

Scale locking locks the scale that you set for the selected viewport. Once the scale is locked, you can continue to modify the geometry in the viewport without affecting the viewport scale. If you turn a viewport's scale locking on, most of the viewing commands, such as , , , , and , no longer function in that viewport.

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**NOTE** Viewport scale locking is also available for nonrectangular viewports. To lock a nonrectangular viewport, you must perform an extra step in the Properties palette to select the viewport object rather than the viewport clipping boundary.

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#### 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.

Standard toolbar

Select the viewport and right-click in the drawing area. Click Properties.

**To turn on scale locking in a layout viewport**

1. In the layout, click the viewport whose scale you want to lock.
2. If necessary, open the Properties palette.
3. In the Properties palette, do one of the following:
   - If you selected a rectangular viewport, select Display Locked, and then click Yes.
   - If you selected a nonrectangular viewport, first click All (2) and select Viewport (1). Then select Display Locked, and click Yes.

   The current viewport's scale is locked. If you change the zoom factor in the viewport, only paper space objects are affected.

Standard toolbar

Select the viewport and right-click in the drawing area. Click Properties.

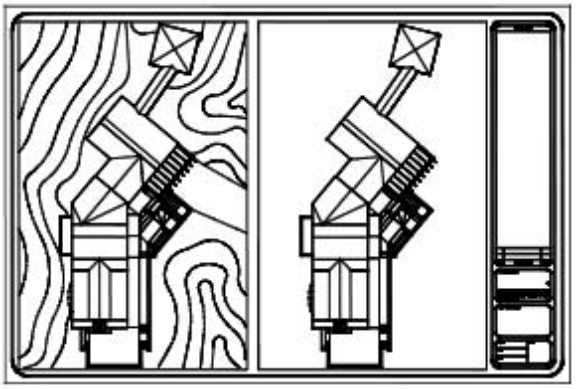
# Control Visibility in Layout Viewports

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 Current VP Freeze to freeze one or more layers in the current layout viewport. To display the Current 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 Format menu ➤ Layer.
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 Current VP Freeze column for one of the selected layers.
5. Click OK.

Layers toolbar

**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 Format menu ➤ Layer.
4. In the Layer Properties Manager, look at the Current VP Freeze column for the Freeze/Thaw in Current Viewports icon.

5. Click OK.

Layers toolbar

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

1. Click a layout tab.
2. Click Format menu ➤ Layer.
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.

Layers toolbar

### 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 Format menu ➤ Layer.
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.

Layers toolbar

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

1. Click a layout tab.
2. Click Format menu ➤ click Layer.
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.

Layers toolbar

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

1. Click a layout tab.

2 Click Format menu ➤ Layer.

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.

Layers toolbar

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## 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 886

**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.

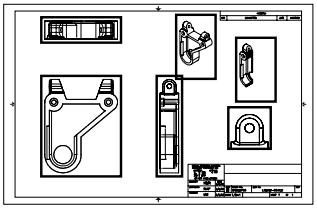
6 In the layout, click File menu ➤ Page Setup.

7. In the Page Setup dialog box, Plot Device tab, select the edited plot style table from the Plot Style Table (Pen Assignments) list.
8. Double-click within the layout viewport that contains the objects whose screening you want to change.
9. Select the objects whose plot style you want to change.
10. Right-click in the drawing area and click Properties.
11. In the Properties palette, use one of the following methods:
    - If you are using named plot style tables, next to Plot Style, select the plot style you edited in the Plot Style Table Editor. If that plot style is not listed, select Other and set Active Plot Style Table to the plot style you edited in the Plot Style Table Editor. Select the edited plot style from the Plot Styles list in the Select Plot Style dialog box.
    - If you are using color-dependent plot style tables, next to Color, select the color whose plot style you edited in the Plot Style Table Editor.

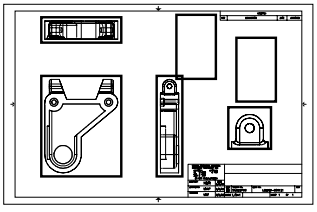
## Turn Layout Viewports On or Off

You can save time by turning some layout viewports off or by limiting the number of active viewports.

Displaying a large number of active layout viewports can affect your system's performance as the content of each layout viewport regenerates. You can save time by turning some layout viewports off or by limiting the number of active viewports. The following illustration shows the effects of turning off two layout viewports.



all viewports on



two viewports off

New layout viewports are turned on by default. If you turn off the layout viewports you aren't using, you can copy layout viewports without waiting for each one to regenerate.

If you don't want to plot a layout viewport, you can turn the layout viewport off.

**To turn viewports on or off using the Properties palette**

1. Make sure you are on a layout tab in paper space.
2. Click the border of the viewport to turn on or off.
3. Right-click, and then click 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.

Standard toolbar

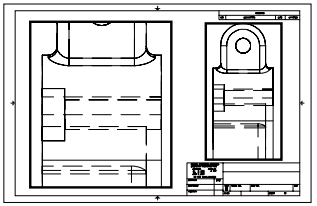
Select the viewport and right-click in the drawing area. Click Display Viewport Objects.

# Scale Linetypes in Layout Viewports

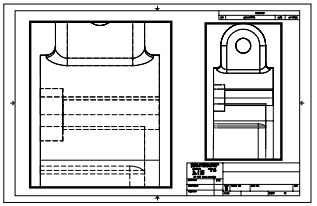
You can scale linetypes in paper space either based on the drawing units of the space in which the object was created or based on the paper space units.

You can set the  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  and . In the following illustration, the pattern of the linetypes in the drawing on the left has been scaled to be the same regardless of the scale of the view. In the drawing on the right, the scale of the linetypes matches the scale of each view.



PSLTSCALE=1, dashes scaled to paper space



PSLTSCALE=0, dashes scaled to space where they were created

**See also:**

"Set the Lineweight Scale for a Layout" on page 855

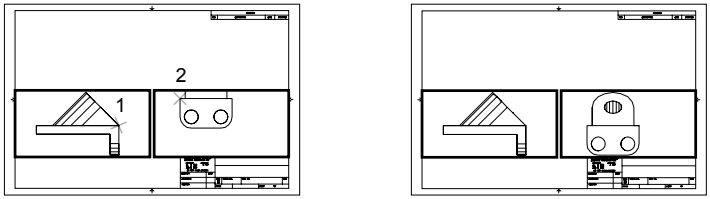
**To scale linetypes globally in paper space**

1. Click Format menu ➤ 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.

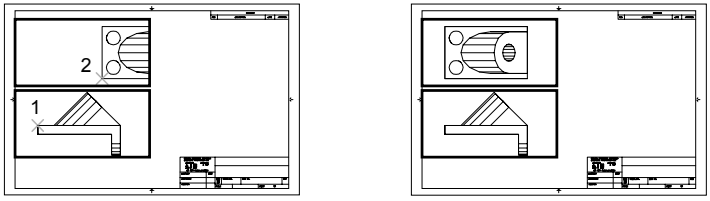
# Align Views in Layout Viewports

You can arrange the elements of your drawing by aligning the view in one layout viewport with the view in another viewport.

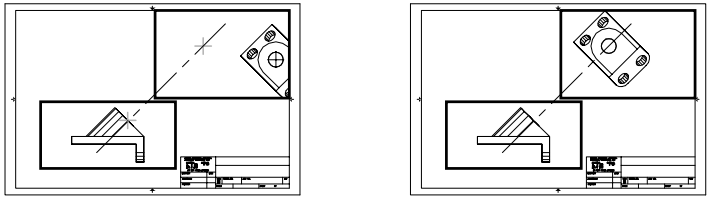
For angled, horizontal, and vertical alignments, you can move each layout viewport relative to distances defined by the model-space geometry displayed.




Horizontal alignments




Vertical alignments



Angled alignments

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

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

1. Make sure you are on a layout tab.
2. Click Draw menu ➤ 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 Modify menu ➤ 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. Choose Object Snap menu ➤ Perpendicular. 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.

Modify toolbar

**To align objects between viewports using MVSETUP**

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

2 Enter **a** (Align).

3 Select one of the following alignments:

- **Horizontal .** Aligns a point in one viewport horizontally with a base point in another viewport.
- **Vertical .** Aligns a point in one viewport vertically with a base point in another viewport.
- **Angled .** Aligns a point in one viewport at a specified distance and angle from a base point in another viewport.

4 Make sure the viewport with the view that is to remain stationary is current. Then specify a base point.

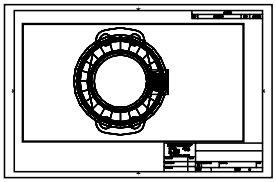
5 Select the viewport with the view you want to realign. Then specify an alignment point in that view.

6 For angled alignments only, specify a distance and displacement angle from the base point to the alignment point in the second viewport.

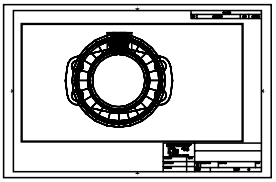
# Rotate Views in Layout Viewports

You can rotate an entire view within a layout viewport by changing the UCS and using the PLAN command.

With the  command, you can rotate the *XY* plane at any angle around the Z-axis. When you enter the PLAN command, the view rotates to match the orientation of the *XY* plane.



original view



rotated view

For a faster method, use the Align and then Rotate View options in the command.

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**NOTE** The  command rotates individual objects only and should not be used to try to rotate a view.

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**To rotate a view by changing the UCS**

1. Make sure you are on a layout tab.
2. Double-click within the viewport whose objects you want to rotate.
3. Make sure that the current UCS is parallel to the plane of rotation (the UCS icon should look normal). If the UCS is not parallel to the plane of rotation, click Tools menu ➤ New UCS ➤ View.
4. Click Tools menu ➤ New UCS ➤ Z. To rotate the view 90 degrees clockwise, enter **90**. To rotate the view 90 degrees counter-clockwise, enter **-90**.
5. Click View menu ➤ 3D Views ➤ Plan View. Click Current UCS.

   The entire view rotates within the viewport. You may need to respecify the scale of the viewport.

UCS toolbar

**To rotate a view using MVSETUP**

1. At the Command prompt, enter **mvsetup**.
2. Enter **a** (Align).
3. Enter **r** to rotate the view.
4. Select the viewport with the view you want to rotate.
5. Specify a base point for the rotation.
6. Specify the rotation angle.

   The entire view rotates within the viewport.

# 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  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  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.

Layouts toolbar

Right-click the layout tab. Click From Template.

### To save a layout template

1. At the Command prompt, enter **layout**.
2. At the prompt, enter **sa** to save the current layout as a template.
3. Enter the name of the layout you are saving.
4. In the Create Drawing File dialog box, enter a name for the drawing template file you are saving.
5. In Files of Type, select Drawing Template File (*.dwt).
6. Click Save.

### To insert a layout using DesignCenter

1. Click Tools menu ➤ 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.

Standard toolbar

# Work with Sheets in a Sheet Set

# 16

With the Sheet Set Manager, you can organize drawing layouts into named sheet sets. The sheets in a sheet set can be transmitted, published, and archived as a unit.

Sheet sets facilitate the organization and management of drawings in a project, and improve the communication in a work group.

## In this chapter

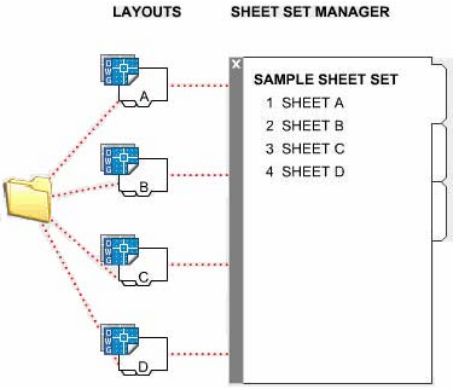

- Quick Start for Sheet Sets
- Understand the Sheet Set Manager Interface
- Create and Manage a Sheet Set
- Publish, Transmit, and Archive Sheet Sets
- Use Sheet Sets in a Team

# Quick Start for Sheet Sets

A sheet set is an organized collection of *sheets* from several drawing files. A sheet is a selected layout from a drawing file.

Sets of drawings are the primary deliverable for most design groups. Sets of drawings communicate the overall design intent of a project and provide the documentation and specifications for the project. However, managing sets of drawings manually can be complicated and time consuming.

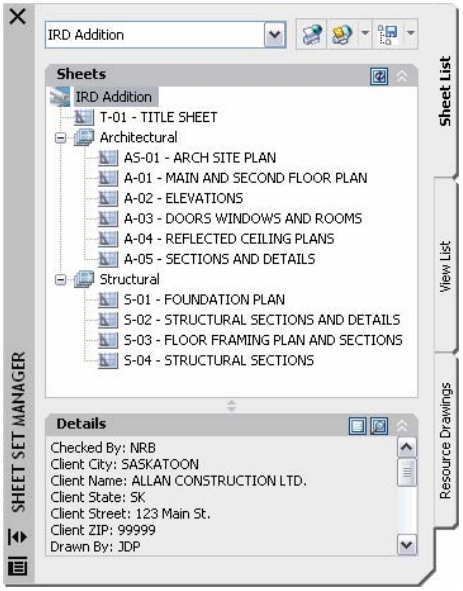
With the Sheet Set Manager, you can manage drawings as *sheet sets*. A sheet set is an organized and named collection of sheets from several drawing files. A sheet is a selected layout from a drawing file. You can import a layout from any drawing into a sheet set as a numbered sheet.




You can manage, transmit, publish, and archive sheet sets as a unit.

# Understand the Sheet Set Manager Interface

Using the controls in the Sheet Set Manager, you can create, organize, and manage sheets in a sheet set.




You can use the following tabs and controls in the Sheet Set Manager:

**Sheet Set Control.** Lists menu options to create a new sheet set, open an existing sheet set, or switch between open sheet sets.

**Sheet List Tab.** Displays an organized list of all sheets in the sheet set. Each sheet in a sheet set is a specified layout in a drawing file.

**View List Tab.** Displays an organized list of all sheet views in the sheet set. Only sheet views created with AutoCAD 2005 and later are listed.

**Resource Drawings Tab.** Lists the paths and folder names for the drawings to be used as resources for the sheet set.

- Click a folder to list the drawing files that are located in that folder.
- Click a drawing file to list the named model space views that are available for placement in the current sheet.
- Double-click a view to open the drawing containing the view.
- Right-click or drag a view to place it in the current sheet.

**Buttons.** Provides convenient access for the most commonly used operations for the currently selected tab.

**Tree View.** Displays the contents of a tab.

**Details or Preview.** Displays either descriptive information or a thumbnail preview of the currently selected item in the tree view.

#### Actions Used in the Tree View

You can use the following actions in the tree view:

- Right-click to access shortcut menus of the operations that are relevant to the currently selected item.
- Double-click items to open them. This is a convenient method for opening drawing files from the Sheet List tab or from the Resource Drawings tab. You also double-click items in the tree view to expand or collapse them.
- Click one or more items to select them for operations such as opening, publishing, or transmitting.
- Click a single item to display descriptive information or a thumbnail preview of a selected sheet, view, or drawing file.
- Drag items within the tree view to reorder them.

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**NOTE** To use the Sheet Set Manager effectively, right-click items in the tree view to access relevant shortcut menus. For access to shortcut menus in the drawing area that are needed for sheet set operations, the Shortcut Menus in Drawing Area must be checked in the Options dialog box, User Preferences tab.

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# Create and Manage a Sheet Set

There are several methods for setting up and organizing a sheet set. You can also include relevant information with a sheet set and its components.

## Create a Sheet Set

You create a sheet set with the Create a Sheet Set wizard. In the wizard, you can either create a sheet set from scratch based on existing drawings or use an example sheet set as a template.

Layouts from specified drawing files are imported into the sheet set. The associations and information that define a sheet set are stored in a sheet set data (DST) file.

When creating a new sheet set using the Create Sheet Set wizard, a new folder is created as the default sheet set storage location. This new folder, named AutoCAD Sheet Sets is located in the My Documents folder. The default

location for the sheet set file can be changed, and it is recommended that the DST file is stored with the project files.

**NOTE** The DST file should be stored in a network location that is accessible to all sheet set users on the network and mapped using the same logical drive. It is strongly recommended that you store the DST and the sheet drawings in the same folder. If an entire sheet set needs to be moved, or a server or folder name changes, the DST file will still be able to locate the sheets using relative path information.

#### Preparation Tasks

Before you begin creating a sheet set, you should complete the following tasks:

- **Consolidate drawing files.** Move the drawing files to be used in the sheet set into a small number of folders. This will simplify sheet set administration.
- **Eliminate multiple layout tabs.** Each drawing you plan to use in the sheet set should have only one layout to be used as a sheet in the sheet set. This is important for access to sheets by multiple users. Only one sheet in each drawing can be open at a time.
- **Create a sheet creation template.** Create or identify a drawing template (DWT) file to be used by the sheet set for creating new sheets. This drawing template file is called the *sheet creation template*. You specify this template file in the Sheet Set Properties dialog box or the Subset Properties dialog box.
- **Create a page setup overrides file.** Create or identify a DWT file to store page setups for plotting and publishing. This file, called the *page setup overrides file*, can be used to apply a single page setup to all sheets in a sheet set, overriding the individual page setups stored in each drawing.

**NOTE** Although it is possible to use several layouts from the same drawing file as separate sheets in a sheet set, it is not recommended. This makes concurrent access to each layout by multiple users impossible. This practice can also reduce your management options and can complicate the organization of your sheet sets.

#### Create a Sheet Set from an Example Sheet Set

In the Create Sheet Set wizard, when you choose to create a sheet set from an example, the example sheet set provides the organizational structure and default settings for the new sheet set. You can also specify that folders are created corresponding to the subset storage paths of the sheet set.

After you create an empty sheet set with this option, you can import layouts or create sheets individually.

### Create a Sheet Set from Existing Drawing Files

In the Create Sheet Set wizard, when you choose to create a sheet set from existing drawing files, you specify one or more folders that contain drawing files. With this option, you can specify that the subset organization for the sheet set duplicates the folder structure of the drawing files. The layouts from these drawings can be imported into the sheet set automatically.

You can easily add more folders containing drawings by clicking the Browse button for each additional folder.

### Backup and Recover Sheet Set Data Files

The data stored in the sheet set data file represents a significant amount of work, so you should take the same care to create backups of DST files as you do for drawing files.

In the unlikely event of DST file corruption or a major user error, the previously saved sheet set data file can be recovered. Every time the sheet set data file is opened, the current sheet set data file is copied to a backup file (DS$). This backup file has the same file name and is located in the same folder as the current sheet set data file.

To recover the previous version of the sheet set data file, first make sure that no one else on your network is working on the sheet set. Then, it is recommended that you copy the existing DST file to another file name. Finally, rename the backup file from the DS$ file extension to the DST file extension.

### To open the Sheet Set Manager

■ Click Tools menu ➤ Sheet Set Manager.

Standard toolbar

### To create a new sheet set

1. To create a new sheet set, do *one* of the following:
   - ■ Click File menu ➤ New Sheet Set.
   - ■ In the Sheet Set Manager, click the Sheet Set control. Click New Sheet Set.
2. Follow the steps in the Create Sheet Set wizard.

Standard toolbar

**To open a sheet set**

1. To open a sheet set, do *one* of the following:
   - Click File menu ➤ Open Sheet Set.
   - In the Sheet Set Manager, click the Sheet Set control. Click Open.
   - Double-click a sheet set data (DST) file.
2. In the Open Sheet Set dialog box, navigate to a folder that contains a DST file. Click the DST file and then click open.

   The Sheet Set Manager displays the sheet set data.

   **NOTE** You can open several sheet sets and then use the Sheet Set control to switch between them.

Standard toolbar

**To close a sheet set**

1. In the Sheet Set Manager, Sheet List tab, right-click the sheet set node (at the top of the list). Click Close Sheet Set.

   The sheet set is no longer displayed in the Sheet Set Manager.

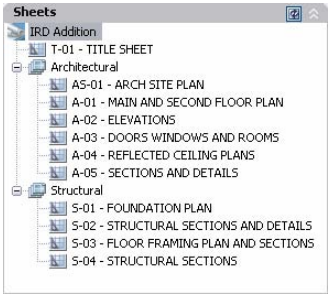
   **NOTE** Close a sheet set to reduce the number of sheet sets listed in the Sheet Set control. However, if you close a sheet set while you still have sheets in the sheet set open, you will not be able to update the Sheet List table and certain fields.

Standard toolbar

# Organize a Sheet Set

You can organize a sheet set by creating a hierarchy of subsets and categories. Sheets can be arranged into collections called subsets. Views can be arranged into collections called categories.

With a large sheet set, you will find it essential to organize sheets and views in the tree view. On the Sheets tab, sheets can be arranged into collections called *subsets*. On the View List tab, views can be arranged into collections called *categories*.




### Use Sheet Subsets

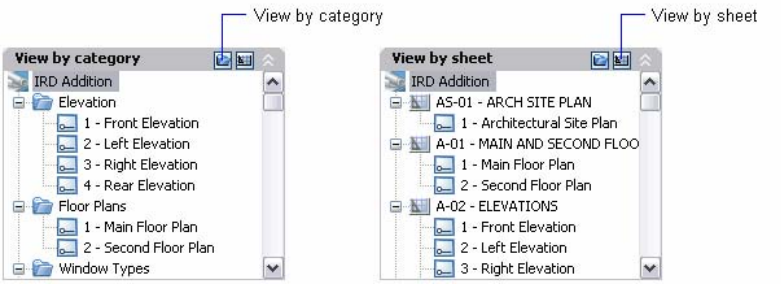
Sheet subsets are often associated with a discipline such as architecture or mechanical design. For example, in architecture, you might use a subset named Structural; and in mechanical design, you might use a subset called Standard Fasteners. In some cases, you might also find it useful to create subsets associated with a review or completion status.

You can nest subsets into other subsets as needed. After you create or import sheets or subsets, you can reorder them by dragging them in the tree view.

### Use View Categories

View categories are often associated with a function. For example, in architecture, you might use a view category called Elevations; and in mechanical design, you might use a view category called Exploded.

You can display views either by category or by the sheet on which they are located.




You can nest categories into other categories as needed. To move a view to a different category, drag them in the tree view or use the Set Category shortcut menu option.

### To reorder sheets in the sheet list

- In the Sheet Set Manager, on the Sheet List tab, drag a sheet up or down the list.

The sheet is relocated above or below its previous location on the sheet list. The sheet may also be moved into or out of a sheet subset.

Standard toolbar

### To create a new subset in the sheet list

1. In the Sheet Set Manager, Sheet List tab, right-click the sheet set node (at the top of the list), or an existing subset. Click New Subset.
2. In the Subset Properties dialog box, under Subset Name, enter the name of the new subset. Click OK.

   You can drag the new subset anywhere on the sheet list, even under other subsets.

   **NOTE** If you want to create a subset under an existing subset, you can right-click the existing subset. On the shortcut menu, click New Subset.

Standard toolbar

**To create a new view category in the view list**

1. In the Sheet Set Manager, View List tab, click the View by Category button.
2. Right-click the sheet set node (at the top of the list). Click New View Category.
3. In the View Category dialog box, under Category Name, enter the name of the new view category.
4. If a list of blocks is displayed, select the callout blocks to be used for the views in this view category. You can add blocks to the list by clicking the Add Blocks button.

Standard toolbar

**To remove a subset from the sheet list**

1. In the Sheet Set Manager, Sheet List tab, drag all sheets out of the subset that you want to remove.
2. Right-click the subset that you want to remove. Click Remove Subset.

Standard toolbar

**To remove a view category from the view list**

1. In the Sheet Set Manager, View List tab, click the View by Category button.
2. Drag all views out of the view category that you want to remove.
3. Right-click the view category that you want to remove. Click Remove Category.

Standard toolbar

# Create and Modify Sheets

There are several options in the Sheet Set Manager for creating sheets and adding views, either through a shortcut menu or one of the tab buttons.

Following are descriptions of common sheet operations. You can access commands by right-clicking an item in the tree view to display the relevant shortcut menu.

- **Import layout as sheet.** After you create a sheet set, you can import one or more layouts from existing drawings. This is a fast method for creating multiple sheets from layouts in several drawings.
- **Create a new sheet.** As an alternative to importing existing layouts, you can create a new sheet. When you place views in this sheet, the drawing files associated with the views are attached as xrefs to the sheet drawing.
- **Modify a sheet.** Double-click a sheet on the Sheet List tab to open a drawing from the sheet set. Use SHIFT or CTRL to select multiple sheets. To review a sheet, use the shortcut menu to open a drawing in read-only mode.

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**NOTE** Modifying a sheet should always be done using an open sheet set in the Sheet Set Manager. This ensures that all data associated with the sheet are updated.

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- **Remove a sheet from a sheet set.** Removing a sheet from a sheet set disassociates the sheet from the sheet set, but does not delete the drawing file or the layout.
- **Reassociate a sheet.** If you move a sheet to a different folder, you should reassociate the sheet to the sheet set with the Sheet Properties dialog box to correct the path. For any relocated sheet drawing, the paths for Expected Layout and Found Layout are displayed in the Sheet Properties dialog box. To reassociate the sheet, click the path in Expected Layout and then click to navigate to the new location of the sheet.

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**NOTE** You can quickly confirm whether a sheet is in the expected folder by looking at Details at the bottom of the Sheet List tab. If the selected sheet is not in the expected location, path information for both Expected Location and Found Location is displayed in Details.

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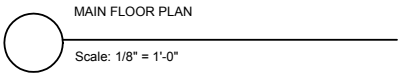
- **Add a view to a sheet.** From the Resource Drawings tab, you can easily add a view to a sheet by placing a named model space view or the entire drawing onto the current sheet.

**NOTE** After creating a named model space view, you must save the drawing to add the view to the Resource Drawings tab. Click Refresh on the Resource Drawings tab to update the Sheet Set Manager tree view.

- **Add label blocks to views.** With the Sheet Set Manager, you can label views and details automatically as you place them. Labels contain data associated with the referenced view.




- **Add callout blocks to views.** *Callout blocks* is the term for the symbols that reference other sheets. Callout blocks have many industry-specific names such as reference tags, detail keys, detail makers, building section keys, and so on. Callout blocks contain data associated with the sheet and view that are referenced.

  **NOTE** When you place a callout block with fields or a view on a sheet, make sure that the current layer is unlocked.

- **Create a title sheet and table of contents.** The first sheet in a sheet set will usually be a title sheet that includes a description of the sheet set and a table that lists all the sheets in the sheet set. You can create this table, called a *sheet list table*, on an open sheet. The table automatically includes all the sheets in the sheet set. You can create a sheet list table from the sheet set level shortcut menu only when a sheet is open.

  **NOTE** For access to shortcut menus in the drawing area that are needed for sheet list table operations, the Shortcut Menus in Drawing Area must be checked in the Options dialog box, User Preferences tab.

#### Create Callout Blocks and Label Blocks (Advanced)

If you create a block to be used as a callout block or label block in a sheet set, you can use a placeholder field to display information such as view title or sheet number. The callout or label block must be defined in a DWG or DWT

file that is specified in the Sheet Set Properties dialog box. Later, you can insert the callout or label block from a shortcut menu on the Views List tab in the Sheet Set Manager.

For the field to display the correct information about a view or sheets on which you later insert it, the field must be included within a block attribute, not text, when you define the block. To create the block attribute definition, insert a placeholder field as the value, select the Preset option, and specify a tag.

**NOTE** If you create your own label blocks and callout blocks, set any attribute definitions to Preset to avoid prompts when placing these blocks in a drawing.

For more information about fields, see "Insert Fields" on page 725.

### Place a Sheet View (Advanced)

The Sheet Set Manager automates and enhances the process for adding views to a sheet. A view on a sheet, called a *sheet view*, consists of several coincident entities: an xref or geometry in model space, a layout viewport on a sheet, and a named view in paper space.

- The sheet view can display model space from a different drawing file. In this case, that drawing is attached as an xref in your current drawing. The layers of that drawing file are displayed only in the sheet view that you create.
- A layout viewport that displays the model space view is created on your current sheet.
- A named view that encompasses the area of the layout viewport is created in paper space.

If you need to remove a sheet view from a sheet, you can delete the layout viewport to remove the view. However, to remove all unused items, you need to detach the xref and delete the named paper space view.

**NOTE** The easiest method for removing a sheet view immediately after placing it is to use .

**See also:**

"Create Multiple-View Drawing Layouts (Paper Space)" on page 203

### To import layouts from existing drawings as sheets

1. In the Sheet Set Manager, Sheet List tab, right-click the sheet set node, a subset node, or a sheet node. Click Import Layout as Sheet.

2. In the Import Layouts as Sheets dialog box, click Browse for Drawings and navigate to the drawing that you want to use.

   To select several drawings, use SHIFT or CTRL when you click on the drawing files.

3. Click the check boxes of the layouts to be imported as sheets in the current sheet set. Click Import Checked.

4. (Optional) On the Sheet List tab, right-click a newly imported sheet. Click Rename & Renumber.

5. (Optional) In the Rename & Renumber Sheet dialog box, make any changes to the sheet number and sheet title. Click OK.

   **NOTE** Changes to the sheet number and sheet title have no effect on the drawing file name unless a file rename option is checked.

Standard toolbar

**To create a new sheet**

1. In the Sheet Set Manager, Sheet List tab, right-click either the sheet set node, a subset node, or a sheet node. Click New Sheet.

2. In the New Sheet dialog box, enter the sheet number and sheet title. Click OK.

   The new sheet is created from the drawing template file specified in Sheet Set Properties for the default new sheet creation template.

   **NOTE** You can change the default folder location for new sheets from the Sheet Set Properties or the Subset Properties dialog box.

Standard toolbar

**To open a drawing file within a sheet set**

1. In the Sheet Set Manager, open a sheet set.

2. On the Sheet List tab, double-click a sheet. To select several drawings, use SHIFT or CTRL when you click on the drawing files.

The drawing file for the sheet is opened. Any changes made to the drawing that are relevant to the sheet list are updated in the sheet set data (DST) file.

Standard toolbar

**To open a drawing file read-only within a sheet set**

1. In the Sheet Set Manager, open a sheet set.
2. On the Sheet List tab, right-click a sheet. Click Open read-only.

Standard toolbar

**To remove a sheet from a sheet set**

1. In the Sheet Set Manager, open a sheet set.
2. On the Sheet List tab, right-click the sheet that you want to remove. Click Remove Sheet

   The specified sheet is removed from the sheet list. The sheet and the drawing file are not deleted, however. The sheet can be added to a different sheet set.

   **NOTE** The DWG file associated with a sheet does not have to be accessible for the sheet to be removed from the sheet set.

Standard toolbar

**To reassociate a sheet in a sheet set**

1. In the Sheet Set Manager, open a sheet set.
2. On the Sheet List tab, open the sheet that you want to reassociate.
3. In the Sheet Set Manager, right-click the sheet. Click Remove Sheet.
4. Save the drawing.

5 In the Sheet Set Manager, right-click the Sheet Set. Click Import Layout as Sheet.

6 In the Import Layout as Sheet dialog box, click Browse for Drawings. Navigate to the drawing that you want to use.

7 Click the check box of the layout to be reassociated as a sheet in the current sheet set. Click Import Checked.

Standard toolbar

**To add a view to a sheet**

1 In the Sheet Set Manager, open a sheet set.

2 On the Sheet List tab, do *one* of the following:

- Double-click a sheet to open it.
- Create a new sheet and open it.

3 On the Resource Drawings tab, click the plus sign (+) next to a folder to list the drawings in the folder.

4 From the list of drawing files, do *one* of the following:

- To add a model view to a sheet, click the plus sign (+) next to a drawing file to list its named model space views. Right-click a model space view.
- To add an entire drawing as a view in a sheet, right-click a drawing file.

5 Click Place on Sheet.

**NOTE** As an alternative to the previous two steps, you can drag a model space view or a drawing from the Resources tab to a sheet.

6 Right-click on the sheet. Click the scale for the sheet view.

7 Specify the insertion point for the sheet view.

The specified view is added to the sheet. If a view label block is defined in the sheet set properties, a view label that displays view-specific information is automatically placed on the sheet.

Standard toolbar

**To add a block to be used for sheet view callouts**

1. In the Sheet Set Manager, open a sheet set.
2. On the View List tab, right-click the sheet set node.
3. In the Sheet Set Properties dialog box, click Callout Blocks. Click the [...] button.
4. In the List of Blocks dialog box, do *one* of the following:
   - Click a block from the list of blocks.
   - Click the Add button and specify a new block to be added to the list.
5. Click OK to close each dialog box.

Standard toolbar

**To change the default block used for sheet view labels**

1. In the Sheet Set Manager, open a sheet set.
2. On the View List tab, right-click the sheet set node.
3. In the Sheet Set Properties dialog box, click Label Block for Views. Click the [...] button.
4. In the Select Block dialog box, specify a new block to be used as the default view label block.
5. Click OK to close each dialog box.

Standard toolbar

**To add a callout block to a sheet**

1. In the Sheet Set Manager, open a sheet set.

2 On the View List tab, right-click the view with which you want to associate a callout. Click Place Callout Block.

3 Specify the insertion point of the callout block.

The callout block is placed on the sheet. The callout block automatically displays view-specific information about the view with which it is associated.

Standard toolbar

**To create a sheet list table on a title sheet**

1 In the Sheet Set Manager, open a sheet set.

2 On the Sheet List tab, double-click the sheet to be used as the title sheet.

3 Right-click the sheet set node. Click Insert Sheet List Table.

4 In the Sheet List Table dialog box, enter the title of the table. Make any formatting changes needed.

5 Click OK.

6 Specify the insertion point of the table.

The sheet list table automatically generates a list of all the sheets in the sheet set.

Standard toolbar

# Include Information with Sheets and Sheet Sets

Sheet sets, subsets, and sheets include several types of information. This information, called properties, includes titles, descriptions, file paths, and custom properties that you define.

**Different Properties for Different Levels (Owners)**

Sheet sets, subsets, and sheets represent different levels of organization, and each of these includes different types of properties. You specify the values for these properties when you create the sheet set, subset, or sheet.

In addition, you can define custom properties for a sheet and a sheet set. The values for custom properties for sheets are typically specific to each sheet. For example, a custom property for a sheet might include the name of the designer. The values for custom properties for a sheet set are typically specific to a project. For example, a custom property for a sheet set might include the contract number.

You cannot create custom properties for subsets.

### View and Edit Properties

You can view and edit properties from the Sheet List tab by right-clicking the name of the sheet set, subset, or sheet. On the shortcut menu, click Properties. The properties and values that are displayed in the Properties dialog box depend on what you selected. You can edit the values of the properties by clicking a value.

#### To edit the properties of a sheet set, subset, sheet, or view category

1. In the Sheet Set Manager, open a sheet set.
2. Do *one* of the following:
   - On the Sheet List tab, right-click the sheet set node, a subset, or a sheet.
   - On the View List tab, right-click a view category.
3. Click Properties.
4. In the dialog box that is displayed, click any field and enter the changes.
5. Click OK.

Standard toolbar

#### To create a custom property for a sheet set

1. In the Sheet Set Manager, open a sheet set.
2. On the Sheet List tab, right-click the sheet set node. Click Properties.
3. In the Sheet Set Properties dialog box, click the Edit Custom Properties button.

4 In the Custom Properties dialog box, do *one* of the following:

- Click Add to create a new custom property for the sheet set. Click the name and the default value to enter the information for these fields. To assign this property to the sheet set, make sure that Sheet Set is selected under Owner.
- Click the name and the default value to change the information for these fields.
- Click a custom property from the list. Click Delete to remove the property.

5 Click OK to close each dialog box.

Standard toolbar

**To create a custom property for a sheet**

1 In the Sheet Set Manager, open a sheet set.

2 On the Sheet List tab, right-click the sheet set node.

3 In the Sheet Set Properties dialog box, click the Edit Custom Properties button.

4 In the Custom Properties dialog box, do *one* of the following:

- Click Add to create a new custom property for the sheet. Click the name and the default value to enter the information for these fields. To assign this property to the sheet, make sure that Sheet is selected under Owner.
- Click the name and the default value to change the information for these fields. With this method, you can change the value for every sheet in the sheet set.
- Click a custom property from the list. Click Delete to remove the property.

5 Click OK to close each dialog box.

Standard toolbar

# Publish, Transmit, and Archive Sheet Sets

After you have organized drawings into a sheet set, you can publish, transmit, and archive the sheet set as a package.

- **Publish a sheet set.** Use the Publish feature to create single and multiple-sheet Design Web Format™ (DWF™ ) files from a sheet set or portion of a sheet set. Alternatively, you can plot the sheet set. For more information, see "Publish a Sheet Set" on page 934. For information about DWF files, see "Plot DWF Files" on page 913.
- **Set options for properties to include in a published DWF file.** You can decide what types of information to reveal in your published DWF files. The types of metadata that you can include are sheet and sheet set properties, block properties and attributes, dynamic block properties and attributes, and properties contained in custom objects. Metadata is only included when you publish to DWF; it is not available when plotting to DWF.
- **Transmit a sheet set.** Package and send a sheet set or a portion of a sheet set over the Internet. For more information, see "Package a Set of Files for Internet Transmission".
- **Archive a sheet set.** Package a sheet set or portion of a sheet set for storage. This is very similar to packaging a transmittal set, except that you specify a folder for the archive and you do not transmit the package. For more information, see the ARCHIVE command.

### Use Page Setups

Page setups provide the settings that are used for publishing and plotting. When you create a sheet set, you specify a drawing template (DWT) file that contains one or more page setups for all new sheets. This DWT file is called the *sheet creation template*.

Another DWT file, called the *page setup overrides* file, contains page setups that can be specified to override the page setups in each sheet. You specify the page setup overrides file in the Sheet Set Properties dialog box.

When you publish a sheet set, you can use the page setups defined in each drawing file, you can use the page setup overrides for all drawing files, or you can publish to a DWF file. For more information about page setups, see "Use Named Page Setups with Sheet Sets" on page 867.

### Save a Selection of Sheets

You can select part of a sheet set for publishing and transmitting. On the Sheet List tab, you can select individual sheets using standard Microsoft® Windows® selection methods—press CTRL or SHIFT when you click items. You can specify all the sheets in a sheet subset by clicking the subset node.

You will likely need to perform operations on the same group of sheets in a sheet set repeatedly. To speed up selecting the sheets and to ensure that the same sheets are selected each time, you can use the Sheet Set Manager to reselect groups of sheets by name. These named groups of sheets are called *sheet selections*. You can create and manage sheet selections by using the Sheet Selections button at the top of the Sheet List tab.

#### To save a sheet selection

1. In the Sheet Set Manager, open a sheet set.
2. On the Sheet List tab, click the sheets and subsets to include in the sheet selection.

   You can use CTRL or SHIFT to specify several items from the list.
3. In the Sheet List Manager, near the upper-right corner, click the Sheet Selections button. Click Create.
4. In the New Sheet Selection dialog box, enter the name of the sheet selection. Click OK.

Sheet Set Manager window

#### To restore a sheet selection

1. In the Sheet Set Manager, open a sheet set.
2. On the Sheet List tab, near the upper-right corner, click the Sheet Selections button. Select the name of the sheet selection that you want to restore.

   The sheet selection is activated and available for publishing and transmitting operations.

Sheet Set Manager window

### To rename or delete a sheet selection

1. In the Sheet Set Manager, open a sheet set.
2. On the Sheet List tab, near the upper-right corner, click the Sheet Selections button. Click Manage.
3. In the Sheet Selections dialog box, select the name of a sheet selection and do *one* of the following:
   - Click Rename to rename the sheet selection. Enter the new name for the sheet selection.
   - Click Delete to remove the sheet selection name from the list. Click Yes to confirm that you want to remove this sheet selection name.
4. Click OK.

Sheet Set Manager window

### To modify an existing sheet selection

**NOTE** You cannot modify a sheet selection directly. Instead, you activate a sheet selection and modify it, delete the name of the sheet selection, and then create a new sheet selection in its place.

1. In the Sheet Set Manager, open a sheet set.
2. On the Sheet List tab, near the upper-right corner, click the Sheet Selections button. Select the name of the sheet selection to modify.
3. Use the CTRL key to add and remove items from the sheet selection.
4. On the Sheet List Manager, near the upper-right corner, click the Sheet Selections button. Click Manage.
5. Click Delete to remove the sheet selection name from the list. Click Yes to confirm that you want to remove this sheet selection name.
6. Click OK to close the Sheet Selections dialog box.
7. On the Sheet List Manager, near the upper-right corner, click the Sheet Selections button. Click Create.
8. In the New Sheet Selection dialog box, enter the name of the sheet selection. Click OK.

Sheet Set Manager window

**To include layer, block, sheet set and sheet information in a published DWF file**

1. In the Sheet Set Manager, Sheet List tab, select the sheet set to publish in a DWF file.
2. Click the Publish button. Click Sheet Set Publish Options.
3. In the Sheet Set Publish Options dialog box, under DWF Data Options, depending on what you want to include in the published DWF file, click any of the following to change the option to "Include."
   - Layer information
   - Sheet set information (attributes you can choose to include are description and custom properties)
   - Sheet information (attributes you can choose to include are sheet title, sheet number, description, sheet set, subset, and sheet custom properties)
   - Block information

   **NOTE** Some sheet properties are always published in the DWF file (no matter what setting you make in the Sheet Set Publish Options dialog box). These are: sheet name, sheet size, author, creator, creation time, and modification time.
4. Click OK.

Sheet Set Manager window

**To use a block template file to include block properties and attributes in a published DWF file**

1. In the Sheet Set Manager, Sheet List tab, select the sheet set to publish in a DWF file.
2. Click the Publish button. Click Sheet Set Publish Options.

3 In the Sheet Set Publish Options dialog box, under DWF Data Options, Block Information, click to display the drop-down list, and select "Include."

4 Under DWF Data Options, Block Template File, select the block template file you want to use or create a new one. Click OK.

The block template file defining the blocks and their properties and attributes will be included in the published DWF file.

Sheet Set Manager window

# Use Sheet Sets in a Team

You can use sheet sets in a team that can involve network access, Internet collaboration, and email transmittal. The team can also include people who use software that does not include Sheet Set Manager.

### Work in a Team That Uses Sheet Set Manager

When you use sheet sets in a team, each member should have network access to the sheet set data (DST) file and the drawing template (DWT) files associated with the sheet set. Each team member can open the sheet set to load the sheet set information from the DST file into the Sheet Set Manager.

Any changes that any team member makes opens the DST file briefly and updates the information stored in that file. When the DST file is opened, a lock icon is displayed next to the sheet set name in the top left corner of the Sheet Set Manager. A green dot in the lock icon indicates that the Sheet Set Manager session on your computer has temporarily locked the DST file; a red dot indicates that the Sheet Set Manager session on a team member's computer has temporarily locked the DST file. Other members of the team can automatically see changes to the sheet set in the Sheet Set Manager tree view.

If each member of the team has access to the sheet set DWT files, new drawing files and their sheets are created using the same drawing template file; page setups for these drawings are also standardized.

**NOTE** If two or more users access the same sheet files through different logical drives on a network, each will in turn be prompted to resave the sheet set using their own logical drive. To avoid unnecessary saving, users should map the same logical drives, if possible.

Status data for sheets in the current sheet set is also available to other team members. This status data is displayed in the tree view and indicates one of the following conditions:

- The sheet is available for editing (a normal sheet icon is displayed).
- The sheet is locked (a lock icon is displayed).
- The sheet is missing or found in an unexpected folder location (a ? icon is displayed).

The active sheets of other team members are automatically polled for status changes; the tree view is updated in your session of the Sheet Set Manager. The polling cycle skips the poll interval in your session when a command is active. To force a sheet status update, click Refresh Sheet Status on the Sheet List tab.

You can click any sheet to display more information in the Details area of the Sheet Set Manager.

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**NOTE** A false lock icon may be displayed if there is a network problem or if the program terminates unexpectedly. If you suspect a problem, click the sheet to display more information.

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#### Work in a Team That Does Not Use Sheet Set Manager

With some limitations, you can use sheets sets in a team with members who do not have network access, or do not have access to the Sheet Set Manager. These team members may be using AutoCAD LT or an older version of AutoCAD. In those circumstances, not all members of the team will have access to the DST file. However, relevant information from the DST file is stored (cached) in each drawing file, and sheet set information, such as custom properties, is preserved when the drawing file is shared by other team members.

After a member of the team changes information in the DST file, the information in several drawing files might be stale and need to be updated. With the sheet set open, update a sheet by opening and saving the sheet. You can update all sheets in a sheet set automatically with the Resave All Sheets option in the sheet set shortcut menu. All drawing files in the sheet set will be saved to the current DWG file format. Drawing files saved in formats earlier than the current DWG file format are updated to the current format regardless of the SAVEAS setting.

---

**NOTE** In a network environment, make sure that all drawing files used in the current sheet set that are opened by other users are closed before performing the Resave All Sheets operation.

---

**To resave all sheets in a sheet set**

1. In the Sheet Set Manager, open a sheet set.
2. On the Sheet List tab, right-click the sheet set node. Click Resave All Sheets.

   All sheets in the sheet set are resaved. This updates the sheet set information that is stored in each drawing file.

Standard toolbar

# Part 5
# Create and Modify Objects

Chapter 17 Control the Properties of Objects
Chapter 18 Use Precision Tools
Chapter 19 Draw Geometric Objects
Chapter 20 Create and Use Blocks (Symbols)
Chapter 21 Change Existing Objects

# Control the Properties of Objects

# 17

You can organize objects in your drawing and control how they are displayed and plotted by changing their properties, which include layer, linetype, color, lineweight, and plot style.

Layers are like transparent overlays on which you organize and group different kinds of drawing information. The objects you create have common properties including colors, linetypes, and lineweights. An object can assume these properties from the layer it is drawn on, or properties can be specifically assigned to individual objects. Color helps you distinguish similar elements in your drawings, while linetypes help you differentiate easily between different drafting elements, such as centerlines or hidden lines. Lineweights represent the size or type of an object through width, enhancing your drawing and increasing legibility. Organizing layers and the objects on layers makes it easier to manage the information in your drawings.

## In this chapter


- Overview of Object Properties
- Display and Change the Properties of Objects
- Copy Properties Between Objects
- Work with Layers
- Work with Colors
- Work with Linetypes
- Control Lineweights
- Control the Display Properties of Certain Objects

# Overview of Object Properties

Every object you draw has properties. Some properties are general and apply to most objects; for example, layer, color, linetype, and plot style. Other properties are object-specific; for example, the properties of a circle include radius and area, and the properties of a line include length and angle.

Most general properties can be assigned to an object by layer or can be assigned to an object directly.

- When a property is set to the value BYLAYER, the object is assigned the same value as the layer on which it is drawn.

  For example, if a line drawn on Layer0 is assigned the color BYLAYER, and Layer0 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 Layer0 is assigned the color Blue, and Layer0 is assigned the color Red, the line is blue.

**See also:**

Control the Color and Linetype Properties in Blocks

# Display and Change the Properties of Objects

You can display and change the current properties for any object in your drawing in the following ways:

- Open the Properties palette and view and change the settings for all properties of the object.
- View and change the settings in the Layer control on the Layers toolbar and the Color, Linetype, Lineweight, and Plot Style controls on the Properties toolbar.
- Use  to view information in the text window.
- Use the  command to display a coordinate location.

### 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 command is on (the default). The exceptions are blocks and attributes, hatches, gradient fills, text, multilines, and xrefs. If you double-click any of these objects, an object-specific dialog box displays instead of the Properties palette.

You can double-click most objects to open the Properties palette when the command is on (the default). The exceptions are blocks and attributes, hatches, text, and xrefs. If you double-click any of these objects, and object-specific dialog box displays instead of the Properties palette.

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**NOTE** The command must be on and the system variable must be on (set to 1, the default) for double-clicking to work.

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**See also:**

Control the Color and Linetype Properties in Blocks
Filter Selection Sets

### To turn Auto-hide on or off for a tool palette

- On the tool palette title bar, click the Auto-hide button.

  If this option is selected, only the tool palette title bar is displayed when the cursor moves outside the tool palette. If this option is clear, the full tool palette is displayed when the cursor moves.

---

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

---

### To remove the description pane at the bottom of the Properties palette

1. On the title bar, click the Properties button.
2. On the shortcut menu, click Description to clear the check mark.

**To display the properties of a single object**

1. Select the object.
2. Right-click in the drawing. Click Properties.

   The Properties palette displays the properties of the selected object.

   Alternatively, you can double-click almost any object to display the Properties palette.

Standard toolbar

**To list database information for several objects**

1. Click Tools menu ➤ Inquiry ➤ List.
2. Select one or more objects and press ENTER.

   The text window displays a report.

Inquiry toolbar

**To display the coordinate values of a point**

1. Click Tools menu ➤ Inquiry ➤ ID Point.
2. Specify the point whose coordinate values you want to identify.

   The *X*, *Y*, and *Z* values are displayed on the command line.

Inquiry toolbar

**To change properties of objects in the Properties palette**

1. Select one or more objects.
2. Right-click in the drawing. Click Properties.

3. In the Properties palette, use the scroll bar next to the title bar to scroll through the list of properties. You can click the arrow at the right of each category to expand or collapse the list.
4. Select the value you want to change; use one of the following methods to change the value:
   - Enter a new value.
   - Click the down arrow at the right and select a value from the list.
   - Click the Pick Point button to use the pointing device to change a coordinate value.
   - Click the QuickCalc calculator button to calculate a new value.
   - Click the left or right arrow to increase or decrease the value.
   - Click the [...] button and change the property value in a dialog box.

   Changes are applied immediately.
5. To undo a change, right-click an empty area in the Properties palette. Click Undo.
6. Press ESC to remove selection.

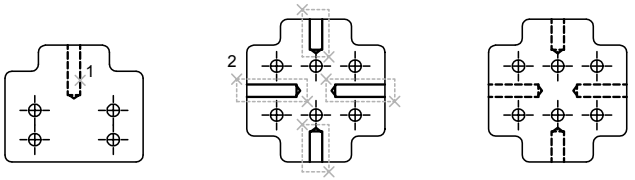
Standard toolbar

# Copy Properties Between Objects

You can copy some or all properties of one object to other objects using Match Properties.

The types of properties that can be copied include, but are not limited to, color, layer, linetype, linetype scale, lineweight, plot style, 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.




source object selected | destination object(s) selected | result

**To copy properties from one object to other objects**

1. Click Standard Toolbar ➤ 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 to apply the selected properties and press ENTER .

Standard toolbar

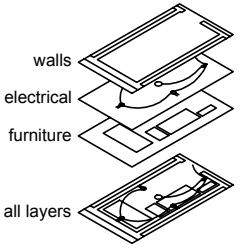
# Work with Layers

Layers are like transparent overlays on which you organize and group different kinds of drawing information.

## Overview of Layers

Layers are used to group information in a drawing by function and to enforce linetype, color, and other standards.

Layers are the equivalent of the overlays used in paper-based drafting. Layers are the primary organizational tool used in drawing. You use layers to group information by function and to enforce linetype, color, and other standards.




By creating layers, you can associate similar types of objects by assigning them to the same layer. For example, you can put construction lines, text, dimensions, and title blocks on separate layers. You can then control

- Whether objects on a layer are visible 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

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.

# 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 which 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 switch layers on and off, the drawing is not regenerated.
- **Freeze/Thaw.** Objects on frozen layers are invisible and do not hide other objects. Thawing one or more layers causes the drawing to be regenerated. Freezing and thawing layers takes more time than turning layers on and off.

In a layout (paper space), you can make some layers visible only in certain viewports.

### Assign a Default Color and Linetype to a Layer

Each layer has associated properties such as color and linetype that are assumed by all objects on that layer. 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.

### Control Whether Objects on a Layer Can Be Modified

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.

### To make a selected layer the current layer

1. On the Layers toolbar, click the Layer control.
2. Click the layer name that you want to use.

**To make the layer of a selected object the current layer**

1. On the Layers toolbar, click Make Object's Layer Current.
2. Select an object.

   The layer of the object you selected is now the current layer.

Layers toolbar

or

**To turn a layer on and off**

1. On the Layers toolbar, click the Layer control.
2. Click the light bulb for the layer name that you want to turn on or off.

   If the light bulb is yellow, the layer is turned on.

**To assign a color to a layer**

1. Click Format menu ➤ Layer.
2. In the Layer Properties Manager, select a layer. Click the color icon.
3. In the Select Color dialog box, select a color.
4. Click Apply to save your changes, or click OK to save and close.

Layers toolbar

**To change the properties of more than one layer**

1. Click Format menu ➤ Layer.
2. In the Layer Properties Manager list view, use one of the following methods to select the layers:

   - Hold down CTRL and select layer names.
   - Right-click. Click Show Filters in Layer List to display a check mark, and then select a layer filter.
3. Click the icons for the properties you want to change.

4 Click Apply to save your changes, or click OK to save and close.

Layers toolbar

**To hide the tree view in the Layer Properties Manager**

■ Right-click in the list view. Clear the Show Filter Tree option.

**To hide a column in the Layer Properties Manager**

■ Drag the right column separator to the left until the column is hidden.

To show the column, drag the column separator to the right.

# Create and Name Layers

By grouping objects into layers, you can control their display and make changes quickly and efficiently.

You can create and name a layer for each conceptual grouping (such as walls or dimensions) and assign common properties to those layers. By grouping objects into layers, you can control their display and make changes quickly and efficiently.

The number of layers you can create in a drawing and the number of objects you can create per layer are virtually unlimited. Layers are usually dedicated to each conceptual grouping (such as walls or dimensions). Layers can have alphanumeric names up to 255 characters long. 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're organizing your own layer scheme, name layers carefully. By using common prefixes to name layers with related drawing components, you can use wild-card characters in layer name filters when you need to find those layers quickly.

---

**NOTE** If you consistently use a specific layering scheme, you can set up a drawing template with layers, linetypes, and colors already assigned. For more information about creating templates, see "Use a Template File to Start a Drawing" on page 105.

---

### 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  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 Format menu ➤ Layer.
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.

   A layer name can include up to 255 characters: letters, digits, and the special characters dollar sign ($), hyphen (-), and underscore (_). Use a reverse quote (`) before other special characters so that the characters are not interpreted as wild-card characters. Layer names cannot include blank spaces.

4 To change the properties, click icons.

When you click Color, Linetype, Lineweight, or Plot Style, a dialog box is displayed.

5 (Optional) Click in the Description column and enter text.

6 Click Apply to save your changes, or click OK to save and close.

Layers toolbar

**To remove an unused layer**

1 On the Layers toolbar, click Layer.

2 In the Layer Properties Manager, select the layer. Click the Delete Layer button.

Layers that have objects assigned to them cannot be removed until those objects are reassigned to a different layer or are deleted. Layers 0 and DEFPOINTS and the current layer cannot be removed.

3 Click Apply to save your changes, or click OK to save and close.

The selected layer is removed.

Layers toolbar

**To purge all unused layers**

1 Click File menu ➤ Drawing Utilities ➤ Purge.

The Purge dialog box displays a tree view of object types with items that can be purged (removed from the drawing).

2 To purge unreferenced layers, use one of the following methods:

- To purge all unreferenced layers, select Layers.
- To purge specific layers, double-click Layers to expand the tree view. Select the layers to be purged.

3 If the item you want to purge is not listed, select View Items You Cannot Purge, select the layer, and read the explanation.

4. You are prompted to confirm each item in the list. If you do not want to confirm each purge, clear the Confirm Each Item to Be Purged option.
5. Click Purge.
6. To confirm the purging of each item, respond to the prompt by choosing Yes or No, or Yes to All if more than one item is selected.
7. Click Close.

# Change Layer Settings and Layer Properties

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

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

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

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

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

### Undo Changes to Layer Settings

You can use Layer Previous to undo changes you make to layer settings. For example, if you freeze several layers and change some of the geometry in a drawing, and then want to thaw the frozen layers, you can do this with a single command without affecting the geometry changes. In another example, if you changed the color and linetype of several layers but later decide you prefer the old properties, you can use Layer Previous to undo the changes and restore the original layer settings.

When you use Layer Previous, it undoes the most recent change or set of changes made using either the Layer control or the Layer Properties Manager. Every change you make to layer settings is tracked and can be undone with

Layer Previous. You can use  to suspend layer property tracking when you don't need it, such as when you run large scripts. There is a modest performance gain in turning off Layer Previous tracking.

Layer Previous does not undo the following changes:

- **Renamed layers.** If you rename a layer and change its properties, Layer Previous restores the original properties but not the original layer name.
- **Deleted layers.** If you delete or purge a layer, using Layer Previous does not restore it.
- **Added layers.** If you add a new layer to a drawing, using Layer Previous does not remove it.

#### To change the layer of one or more objects

1. Select the objects whose layer you want to change.
2. On the Layers toolbar, click the Layer control.
3. Select the layer that you want to assign to the objects.
4. Press  ESC  to remove selection.

Layers toolbar

#### To change the default linetype assigned to a layer

1. Click Format menu ➤ Layer.
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 us. Click OK to return to the Layer Properties Manager.

5 Click Apply to save your changes, or click OK to save and close.

Layers toolbar

### To rename a layer

1 Click Format menu ➤ Layer.

2 In the Layer Properties Manager, select a layer. Click the name or press F2 .

3 Enter a new name.

4 Click Apply to save your changes, or click OK to save and close.

Layers toolbar

### To rename more than one layer

1 On the command line, enter **rename**.

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

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

4 In Rename To, enter the new name using wild-card characters; for example, enter **s_***.

Results for this example are as follows: the layers STAIR$LEVEL-1, STAIR$LEVEL-2, STAIR$LEVEL-3 are renamed S_LEVEL-1, S_LEVEL-2, S_LEVEL-3.

5 Click Rename To to apply changes and continue, or click OK.

### To undo changes to layer settings

■ On the Layers toolbar, click Layer Previous.

The command line displays the message “Restored previous layer status.”

**NOTE** LAYERP does not restore deleted or purged layers and does not remove a layer that was added. If you rename a layer and change its properties, Layer Previous restores the original properties but not the original layer name.

Layers toolbar

**To set Layer Previous tracking on or off**

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

   The current Layer Previous tracking status is displayed.

2. Enter **on** to turn on Layer Previous tracking of layer settings, or enter **off** to turn off tracking.

# 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 filters:** 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 filters:** Includes the layers that are put into the filter when you define it, regardless of their names or properties.

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. Three default filters are displayed:

- **All.** Displays all the layers in the current drawing.
- **All Used.** Displays all the layers on which objects in the current drawing are drawn.

- **Xref.** If xrefs are attached to the drawing, displays all the layers being referenced from other drawings.

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 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 current 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.

### Invert a Layer Filter

You can also invert a layer filter. For example, if all the site plan information in a drawing is contained in multiple layers that include the word *site* as part of the layer name, you can display all information except site plan information by first creating a filter definition that filters layers by name (***site**) and then using the Invert Filter option.

### Sort Layers

Once you have created layers, you can sort them by name or other properties. In the Layer Properties Manager, click the column heading to sort layers by the property in that column. Layer names can be sorted in ascending or descending alphabetical order.

### Wild-Card Characters

You can use wild-card characters to sort layers by name.

| Character | Definition |
|---|---|
| # (Pound) | Matches any numeric digit |
| @ (At) | Matches any alphabetic character |
| . (Period) | Matches any nonalphanumeric character |
| * (Asterisk) | Matches any string and can be used anywhere in the search string |
| ? (Question mark) | Matches any single character; for example, ?BC matches ABC, 3BC, and so on |
| ~ (Tilde) | Matches anything but the pattern; for example; ~*AB*matches all strings that don't contain AB |
| [ ] | Matches any one of the characters enclosed; for example, [AB]C matches AC and BC |
| [~] | Matches any character not enclosed; for example, [~AB]C matches XC but not AC |
| [-] | Specifies a range for a single character; for example, [A-G]C matches AC, BC, and so on to GC, but not HC |

| Character | Definition |
| --- | --- |
| ` (Reverse quote) | Reads the next character literally; for example, `~AB matches ~AB |

**NOTE** If you want to use a wild-card character in the name of a named object, precede the character with a reverse quote (`) so that it is not interpreted as a wild-card character.

**To quickly filter the display of layers by name**

1. Click Format menu ➤ Layer.
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 Format menu ➤ Layer.
2. In the Layer Properties Manager, click the New Property Filter button.
3. In the Layer Filter Properties dialog box, enter a name for the filter.
4. Under Filter Definition, set the layer properties that you want to use to define the filter.

   - To filter by name, use wild-card characters.
   - To filter by property, click in the column for the property you want.

     Some properties display a dialog box when you click the [...] button.
   - To select more than one value for a property, right-click the row in the filter definition. Click Duplicate Row. Select another value for that property in the next row.

     For example, the definition for a filter that displays only layers that are on and are either yellow or red has two rows. The first row of the filter definition has the On icon and red. The second row has the On icon and yellow.

5 Click Apply to save your changes, or click OK to save and close.

Layers toolbar

**To filter the display of layer names by selecting layers**

1 On the Layers toolbar, click Layer Properties Manager.

2 In the Layer Properties Manager, click the New Group Filter button.

A new layer group filter named GROUP FILTER1 is created in the tree view.

3 Enter a name for the filter.

4 In the tree view, click All or one of the other nodes to display layers in the list view.

5 In the list view, select the layers you want to add to the filter, and drag them to the filter name in the tree view.

6 Click Apply to save your changes, or click OK to save and close.

Layers toolbar

**To nest a layer filter under another layer filter**

1 On the Layers toolbar, click Layer Properties Manager.

2 In the Layer Properties Manager tree view, select a layer filter.

- A new layer property filter can be nested under a group filter or another property filter.
- A new layer group filter can be nested only under another group filter.

3 Right-click. Click New Properties Filter or New Group Filter.

4 Use one of the following methods:

- For a new property filter, the Layer Properties Filter dialog box is displayed. Under Filter Definition, set the layer properties that you want to use to define the filter. Click OK.

- For a new group filter, a filter is added to the tree view. Rename it, select the parent filter to display its layers in the list view, and drag layers from the list view to the new layer group filter.

5 Click Apply to save your changes, or click OK to save and close.

Layers toolbar

**To sort the layer list in the Layer Properties Manager**

- Click any column heading.

To reverse the sorting order, click a second time.

Layers toolbar

**To remove a layer from a layer group filter**

1 On the Layers toolbar, click Layer Properties Manager.

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.

Layers toolbar

# Save and Restore Layer Settings

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.

Using the Layer States Manager, you can export named layer states to an LAS file for use in other drawings. The layer states of external references cannot be exported.

### Restore Layer Settings

When you restore a named layer state, the layer settings (layer states and layer properties) that were specified when the layer state was saved are restored by default. Because all the layer settings are saved in the named layer state, you can specify different settings at the time you restore. All layer settings that are not selected to be restored remain unchanged.

In addition, the layer that was current when the named layer state was saved is made current. If that layer no longer exists, the current layer does not change.

In addition to restoring named layer states, you can import named layer states that were previously exported to LAS files.

#### To save layer settings in a named layer state

1. Click Format menu ➤ Layer.
2. In the Layer Properties Manager, click the Layer States Manager button.
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.
5. (Optional) Add a description.
6. Click OK.
7. In the Layer States Manager, select the layer settings to restore by default.
8. (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.

9 Click Close to exit the Layer States Manager. Click OK to exit the Layer Properties Manager.

Layers toolbar

**To export saved layer settings**

1 Click Format menu ➤ Layer.

2 In the Layer Properties Manager, click the Layer States Manager button.

3 In the Layer States Manager, select a named layer state. Click Export.

4 In the Export Layer State dialog box, enter a file name. Specify a location for the layer state (LAS) file.

The *.las* file name extension is added automatically.

5 Click Save to exit the dialog box.

6 Click Close to exit the Layer States Manager. Click OK to exit the Layer Properties Manager.

Layers toolbar

**To restore layer settings**

1 Click Format menu ➤ Layer.

2 In the Layer Properties Manager, click the Layer States Manager button.

3 In the Layer States Manager, select a named layer state.

4 Select the settings to restore. Click Restore.

The Layer States Manager closes.

5 Click OK.

Layers toolbar

**To modify a named layer state**

1. Click Format menu ➤ Layer.
2. In the Layer Properties Manager, click the Layer States Manager button.
3. In the Layer States Manager, select the named layer state that you want to edit, rename, or delete.
4. Use one of the following methods:
   - To rename the named layer state, click the name and enter a new name.
   - To edit the description, click the description and enter new text.
   - To delete the saved layer state, click Delete.
   - To change which layer settings to restore by default, select or clear layer settings.
5. Click Close to exit the Layer States Manager. Click OK to exit the Layer Properties Manager.

Layers toolbar

**To import saved layer settings from another drawing**

1. Click Format menu ➤ Layer.
2. In the Layer Properties Manager, click the Layer States Manager button.
3. In the Layer States Manager, click Import.
4. In the Import Layer State dialog box, select a file name with an *.las* file name extension. 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. Click OK to exit the Layer Properties Manager.

Layers toolbar

# Work with Colors

Color helps to group objects visually. You can assign colors to object by layer or individually.

## Set the Current Color

You can use color to help you identify objects visually. You can assign the color of an object either by layer or by specifying its color explicitly, independent of layer.

You can assign the color of an object either by layer or by specifying its color explicitly, independent of layer. Assigning colors by layer makes it easy to identify each layer within your drawing. Assigning colors explicitly provides additional distinctions between objects on the same layer. Color is also used as a way to indicate lineweight for color-dependent plotting.

You can use a variety of color palettes when assigning color to objects, including

- AutoCAD Color Index (ACI)
- True Color® , PANTONE®
- RAL™ Classic and RAL Design color books
- DIC® Color Guide
- Colors from imported color books.

#### ACI Colors

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

#### True Colors

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

of the color; with the HSL color model, you can specify the hue, saturation, and luminance aspects of the color.

### Color Books

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

---

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

---

You install color books on your system by using the Files tab in the Options dialog box. Once a color book is loaded, you can select colors from the color book and apply them to objects in your drawings.

All objects are created using the current color, which is displayed in the Color control on the Properties toolbar. You can also set the current color with the Color control or the Select Color dialog box.

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

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

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

1. On the Properties toolbar, click the Color control.
2. 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 On the Properties toolbar, click the Color control.

2 Click Select Color to display the Select Color dialog box.

3 In the Select Color dialog box, True Color tab, do one of the following:

- Select the HSL color model in the Color Model box. Specify a color by entering a color value in the Color box or by specifying values in the Hue, Saturation, and Luminance boxes.
- Select the RGB color model in the Color Model box. Specify a color by entering a color value in the Color box or by specifying values in the Red, Green, and Blue boxes.

4 Click OK.

The Color control displays the current color.

### To set a color from a color book for all new objects

1 On the Properties toolbar, click the Color control.

2 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.

## 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, or a true color.

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.

**To change the layer of an object**

1. Select the objects whose layer you want to change.
2. On the Layers toolbar, click the Layer control.
3. Select the layer that you want to assign to the objects.

Layers toolbar

**To change the color assigned to a layer**

1. On the Layers toolbar, click the Layer Properties Manager button.
2. In the Layer Properties Manager, click the color you want to change.
3. In the Select Color dialog box, 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.

Layers toolbar

**To change the color of objects, overriding the layer's color**

1 Select the objects whose color you want to change.

2 Right-click one of the objects. 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.

# Use Color Books

When assigning colors to objects, you can choose colors from color books that are loaded on your system. You can choose from a wide range of custom colors when using color books. Color books include third-party or user-defined files that contain named color swatches. These colors can be used to enhance presentation drawings as well as to optimize the variety of color used in your

drawings. You can apply color book colors to objects in your drawings by using the Color Books tab in the Select Color dialog box.

### Install Color Books

Color book files must contain an *.acb* file extension in order to be recognized by this program. To access color book colors from the Select Color dialog box, you must first copy your color book files to a specified color book location. On the Files tab of the Options dialog box, you can define the path where color book files are stored. Multiple locations can be defined for the color book path. These locations are saved in your user profile.

After loading a color book on your system, to access the new colors, you need to close the Select Color dialog box and then open it again. The new color book is displayed in the Color Book drop-down list on the Color Books tab. Once you have loaded a color book, you can apply any colors that are defined in the book to objects in your drawing.

### Browse Color Books

Color books are organized alphabetically into pages that you can browse through. A page holds up to ten colors. If the color book you are browsing through is not organized into pages, the colors are arranged into pages, with each page containing up to seven colors.

#### To install a color book

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Files tab, click Color Book Locations.
3. Click Add to add a color book location.
4. Enter the new location in the blank path box.
5. Click OK.

#### To search for color swatches within a color book

1. Click Format menu ➤ Color.
2. In the 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.
3. 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.

4 Click OK to apply the color.

**To change the default location of color book files**

1 Click Tools menu ➤ Options.

2 In the Options dialog box, Files tab, double-click Color Book Locations.

3 Select the following default color book location:
*C:\Program Files\[current AutoCAD release number]\support\color*

4 Click the location name to edit the path.

5 Enter the new location in the path box.

6 Click OK.

**To define multiple folders for the color book path**

1 Click Tools menu ➤ Options.

2 In the Options dialog box, Files tab, click Color Book Locations.

3 Click Add to add a color book location.

4 Enter the new location in the blank path box.

5 Click OK.

# Work with Linetypes

You can use linetypes to distinguish objects from one another visually and make your drawing easier to read.

## Overview of Linetypes

A linetype is a repeating pattern of dashes, dots, and blank spaces displayed in a line or a curve. You assign linetypes to objects either by layer or by specifying the linetype explicitly, independent of layers.

In addition to choosing a linetype, you can set its scale to control the size of the dashes and spaces, and you can create your own custom linetypes.

**NOTE** These linetypes should not be confused with the hardware linetypes provided by some plotters. The two types of dashed lines produce similar results.

Do not use both types at the same time, however, because the results can be unpredictable.

**See also:**

**"Custom Linetypes" in the Customization Guide**

# 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  or by deleting the linetype from the Linetype Manager. BYBLOCK, BYLAYER, and CONTINUOUS linetypes cannot be removed.

### To load a linetype

1. Click Format menu ➤ Linetype.
2. In the Linetype Manager, 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. On the Properties toolbar, click the Linetype control.

   All loaded linetypes are displayed in the list. If necessary, use the scroll bar to see the whole list.
2. Click anywhere outside the box to close it.

### To list the linetypes in a linetype definition file

1. Click Format menu ➤ Linetype.
2. In the Linetype Manager, click Load.
3. In the Load or Reload Linetypes dialog box, click File.
4. In the Select Linetype File dialog box, select an LIN (linetype definition) file whose linetypes you want to lit. Click Open.

   The dialog box displays the linetype definitions stored in the selected LIN file.
5. In the Load or Reload Linetypes dialog box, click Cancel.
6. Click Cancel to close the Linetype Manager.

### To unload an unused linetype

1. Click Format menu ➤ Linetype.
2. In the Linetype Manager, 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 File menu ➤ Drawing Utilities ➤ Purge.

   The Purge dialog box displays a tree view of object types with items that can be purged.
2. To purge unreferenced linetypes, use one of the following methods:

   - To purge all unreferenced linetypes, select Linetypes.
   - To purge specific linetypes, double-click Linetypes to expand the tree view. Then select the linetypes to be purged.

   If the item you want to purge is not listed, select View Items You Cannot Purge.

3 You are prompted to confirm each item in the list. If you do not want to confirm each purge, clear the Confirm Each Item to Be Purged option.

4 Click Purge.

To confirm the purging of each item, respond to the prompt by choosing Yes or No, or Yes to All if more than one item is selected.

5 Click Close.

# 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 Format menu ➤ Linetype.

2 If you need to load additional linetypes, click Load, select one or more linetype. Click OK.

You can hold down CTRL to select several linetypes or SHIFT to select a range of linetypes.

3 In the Linetype Manager, do one of the following:

- Select a linetype and select Current to draw all new objects with that linetype.
- Select BYLAYER to draw new objects in the linetype assigned to the current layer.

- Select BYBLOCK to draw new objects in the current linetype until they are grouped into a block. When the block is inserted into a drawing, the objects in the block acquire the current linetype setting.

4 Click OK.

---

**NOTE** The Linetype control displays the current linetype. If the linetype you want to use is already loaded, you can click the Linetype control and click the linetype to make it current.

---

**To change the linetype assigned to a layer**

1 On the Layers toolbar, click the Layer Properties Manager button.

2 In the Layer Properties Manager, select the linetype name you want to change.

3 In the Select Linetype dialog box, select the linetype you want. Click OK.

4 Click OK again.

Layers toolbar

# Change the Linetype of an Object

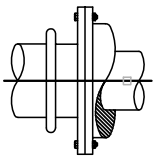
You can change the linetype of an object by reassigning it to another layer, by changing the linetype of the layer the object is on, or by specifying a linetype for the object explicitly.

You have three choices for changing the linetype of an object:

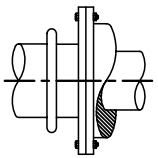
- Reassign the object to another layer with a different linetype. If an object's linetype is set to BYLAYER, and you reassign the object to a different layer, it acquires its linetype from the new layer.
- Change the linetype assigned to the layer that the object is on. If an object's linetype is set to BYLAYER, it acquires the linetype of its layer. When you change the linetype assigned to a layer, all objects on that layer assigned the BYLAYER linetype are updated automatically.
- Specify a linetype for an object to override the layer's linetype. You can specify the linetype of each object explicitly. If you want to override the layer-determined linetype of an object with a different one, change an

existing object's linetype from BYLAYER to a specific linetype, such as DASHED.

If you want to set a specific linetype for all subsequently created objects, change the current linetype setting on the Properties toolbar from BYLAYER to a specific linetype.



object selected



result-continuous linetype changed to center

**To change the layer of an object**

1. Select the objects whose layer you want to change.
2. On the Layers toolbar, click the Layer control.
3. Select the layer that you want to assign to the objects.

Layers toolbar

**To change the default linetype assigned to a layer**

1. On the Layers toolbar, click the Layer Properties Manager button.
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.

Layers toolbar

**To change the linetype of an object, overriding the layer's linetype**

1. Select the objects whose linetype you want to change.
2. On the Properties toolbar, click the Linetype control.
3. Choose the linetype that you want to assign to the 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  system variable, which changes the linetype scale globally for both new and existing objects.
- The Current Object Scale value controls the  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 .

**To change the linetype scale of selected objects**

1. Select the objects whose linetype scale you want to change.
2. Right-click one of the objects. Click Properties.
3. In the Properties palette, select Linetype Scale and enter the new value.

Standard toolbar

**To set the linetype scale for new objects**

1. Click Format menu ➤ Linetype.
2. In the Linetype Manager, click Show Details to expand the dialog box.
3. Enter a new value for Current Object Scale.
4. Click OK.

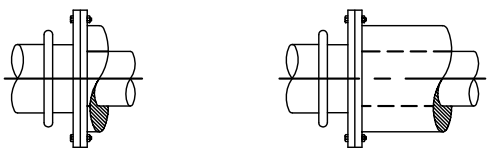
**To change linetype scale globally**

1. Click Format menu ➤ 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.

# Display Linetypes on Short Segments and Polylines

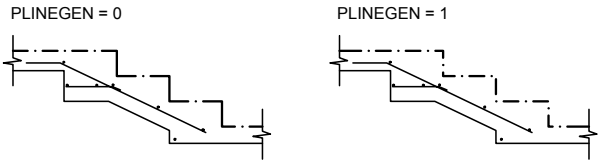
You can center the pattern of a linetype on each segment of a polyline, and you can control how the linetype is displayed on short segments.

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 299.

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  system variable.




**To set the linetype display for all new polylines**

1. At the Command prompt, enter **plinegen**.
2. Enter **1** to make the linetype pattern continue throughout the entire length of two-dimensional polylines, or enter **0** to center the linetype pattern on each segment.

**To change the linetype display of existing polylines**

1. Double-click the polyline whose linetype display you want to change.
2. In the Properties palette, click Linetype Generation and select Enabled or Disabled.

Standard toolbar

# Control Lineweights

You can further differentiate objects in your drawing by controlling their lineweights both in the drawing display and in plotting.

## Overview of Lineweights

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.

All objects except TrueType fonts, raster images, points, and solid fills (2D solids) can display lineweight. In a plan view, wide polylines override any width value applied by a lineweight. Wide polylines display lineweights only when the polyline is being viewed in a view other than Plan. 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  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 on the command line 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 command, by right-clicking the Lwt button on the status bar and choosing Settings, or by choosing Lineweight Settings on the Display tab in the Options dialog box.

**See also:**

"Draw Polylines" on page 391

### To assign width to polylines

1. Click Draw menu ➤ 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.

Draw toolbar

**To assign a lineweight to a layer**

1 Click the Format menu ➤ Layer.

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 Format menu ➤ Lineweight.

2 In the Lineweight Settings dialog box, under Adjust Display Scale, move the slider to change the scale.

3 Click OK.

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# Display Lineweights

Lineweights 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.

### Display Lineweights in Model Space

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.

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  to set their display scale. Changing the display scale does not affect the lineweight plotting value.

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. On the Model tab, you can turn the display of lineweights on or off by clicking Lwt on the status bar.

### 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.

In layouts, 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.

### To display lineweights

Use one of the following methods:

- Toggle 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 to turn lineweight display off or on.

## Set the Current Lineweight

The current lineweight is the lineweight used for any objects you draw until you make another lineweight current.

All objects are created using the current lineweight, which is displayed in the Lineweight control on the Properties toolbar. You can also set the current lineweight with the Lineweight control.

If the current lineweight is set to BYLAYER, objects are created with the lineweight assigned to the current layer.

If the current lineweight is set to BYBLOCK, objects are created using the default lineweight setting until the objects are grouped into a block. When the block is inserted into the drawing, it acquires the current lineweight setting.

If you do not want the current lineweight to be the lineweight assigned to the current layer, you can specify a different lineweight explicitly.

Objects in drawings created in an earlier release of AutoCAD are assigned the lineweight value of BYLAYER, and all layers are set to DEFAULT. Lineweight assigned to objects is displayed as a solid fill drawn in the object's assigned color.

**To make a lineweight current for creating objects**

1. Click Format menu ➤ Lineweight.
2. In the Lineweight Settings dialog box, select a lineweight.
3. Click OK.

**NOTE** To display the lineweight at its current setting, the Display Lineweight option must be selected in the Display Lineweight dialog box.

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# 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.

**To change the line width of a polyline, donut, rectangle, or polygon**

1. Click Modify menu ➤ Object ➤ Polyline.
2. Select one or more polyline objects.
3. Enter **w** (Width) and enter a new width for all segments.
4. Press ENTER to end the command.

Modify II toolbar

# Control the Display Properties of Certain Objects

You can control how overlapping objects and certain other objects are displayed and plotted.

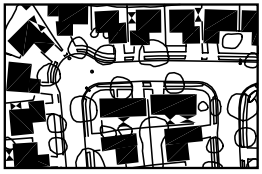
## Control the Display of Polylines, Hatches, Gradient Fills, Lineweights, and Text

You can simplify the display of certain kinds of objects in order to speed performance.

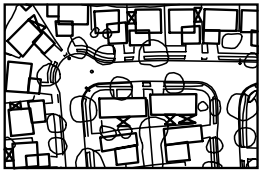
Display performance is improved when wide polylines and donuts, solid-filled polygons (two-dimensional solids), hatches, gradient fills, and text are displayed in simplified form. Simplified display also increases the speed of creating test plots.

### Turn Off Solid Fill

When you turn off Fill mode, wide polylines, solid-filled polygons, gradient fill, and hatches are displayed in outline form. Except for patterned hatches and gradient fills, solid fill is automatically turned off for hidden view and nonplan views in three dimensions.



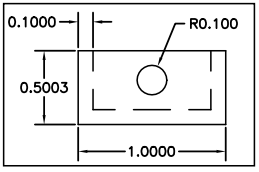
Fill mode on



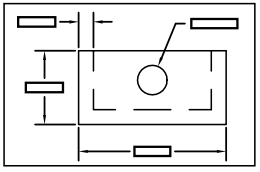
Fill mode off

### Use Quick Text

When you turn on Quick Text mode in drawings that contain a lot of text using complex fonts, only a rectangular frame defining the text is displayed or plotted.




Quick Text mode off



Quick Text mode on

### Turn Off Lineweights

Any lineweight width that is represented by more than one pixel may slow down performance. If you want to improve display performance, turn lineweights off. You can turn lineweights on and off by choosing the Lwt button on the status bar or by using the Lineweight Settings dialog box. Lineweights are always plotted at their real-world value whether their display is turned on or off.

### Update the Display

New objects automatically use the current settings for displays of solid fill and text. Except for lineweights, to update the display of existing objects using these settings, you must use .

**See also:**

"Use Layers to Manage Complexity" on page 269
"Display Lineweights" on page 303
"Use TrueType Fonts" on page 735

### 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.

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### 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.

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### To turn lineweights on or off

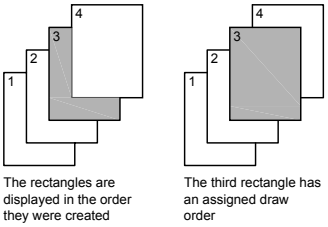
1. Click Format menu ➤ Lineweight.
2. In the Lineweight Settings dialog box, select or clear Display Lineweight.
3. Click OK.

# 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  to change the draw order (which is the display and plotting order) of any objects.  changes the draw order of all text and dimensions 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. On the Draw Order toolbar, click one of the draw order options.
2. Select the object(s) whose draw order you want to modify and press ENTER.
3. Select the reference object(s) and press  ENTER. (This step is necessary only for the Bring Above Objects and Send Under Objects options.)

Tools menu Draw Order

Select an object and right-click. Click Draw Order

# Use Precision Tools

# 18

You can use a variety of precision drawing tools to help you produce accurate drawings quickly and without performing tedious calculations.

## In this chapter


- Use Coordinates and Coordinate Systems
- Use Dynamic Input
- Snap to Locations on Objects (Object Snaps)
- Restrict Cursor Movement
- Combine or Offset Points and Coordinates
- Specify Distances
- Extract Geometric Information from Objects
- Use a Calculator

# Use Coordinates and Coordinate Systems

For precise coordinate input, you can use several coordinate system entry methods. You can also use a movable coordinate system, the user coordinate system, for convenient coordinate entry and to establish drawing planes.

## 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 on the command line. 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 construction plane. The construction plane is similar to a flat sheet of grid paper. The *X* value of a Cartesian coordinate specifies horizontal distance, and the *Y* value specifies vertical distance. The origin point (0,0) indicates where the two axes intersect.

Polar coordinates use a distance and an angle to locate a point. With both Cartesian and polar coordinates, you can enter absolute coordinates based on the origin (0,0), or relative coordinates based on the last point specified.

Another method of entering a relative coordinate is by moving the cursor to specify a direction and then entering a distance directly. This method is called direct distance entry.

You can enter coordinates in scientific, decimal, engineering, architectural, or fractional notation. You can enter angles in grads, radians, surveyor's units, or degrees, minutes, and seconds. The UNITS command controls unit format.

### Display Coordinates on the Status Bar

The current cursor location is displayed as a coordinate value on the status bar.

There are three types of coordinate display: static, dynamic, and distance and angle.

- **Static display**. Updates only when you specify a point.
- **Dynamic display.** Updates as you move the cursor.
- **Distance and angle display.** Updates the relative distance *(distance<angle)* as you move the cursor. This option is available only when you draw lines or other objects that prompt for more than one point.

#### To display the coordinate values of a point

1. Click Tools menu ➤ Inquiry ➤ ID Point.
2. Select the location you want to identify.

   The *X,Y,Z* coordinate values are displayed on the command line.

Inquiry toolbar

#### To visually locate a point

1. Click Tools menu ➤ Inquiry ➤ ID Point.
2. On the command line, 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.

Inquiry toolbar

#### 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 F6 or CTRL+D.
- Set the COORDS system variable to 0 for static display, 1 for dynamic display, or 2 for distance and angle display.

# 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.

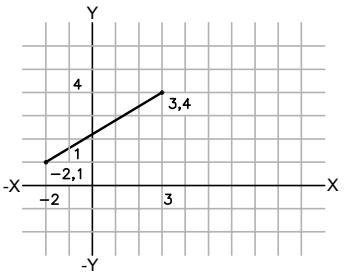
To use Cartesian coordinates to specify a point, enter an *X* value and a *Y* value separated by a comma *(X,Y)*. The *X* value is the positive or negative distance, in units, along the horizontal axis. The *Y* value is the positive or negative distance, in units, along the vertical axis.

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 335.

The following example draws a line beginning at an *X* value of –2, a *Y* value of 1, and an endpoint at 3,4. Enter the following in the tooltip:

Command: **line**
From point: **#–2,1**
To point: **#3,4**
The line is located as follows:

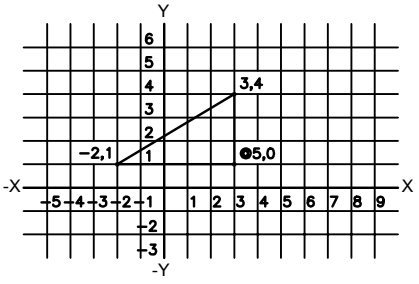


Relative coordinates are based on the last point entered. Use relative coordinates when you know the location of a point in relation to the previous point.

To specify relative coordinates, precede the coordinate values with an @ sign. For example, entering **@3,4** specifies a point 3 units along the *X* axis and 4 units along the *Y* axis from the last point specified.

The following example draws the sides of a triangle. The first side is a line starting at the absolute coordinates -2,1 and ending at a point 5 units in the *X* direction and 0 units in the *Y* direction. The second side is a line starting at the endpoint of the first line and ending at a point 0 units in the *X* direction and 3 units in the *Y* direction. The final line segment uses relative coordinates to return to the starting point.

Command: **line**
From point: **#-2,1**
To point: **@5,0**
To point: **@0,3**
To point: **@-5,-3**




**To enter absolute Cartesian coordinates (2D)**

At a prompt for a point, enter coordinates in the tooltip using the following format:

**#** *x* **,** *y*

If Dynamic Input is turned off, enter coordinates on the command line using the following format:

*x* **,** *y*

**To enter relative Cartesian coordinates (2D)**

At a prompt for a point, enter coordinates using the following format:

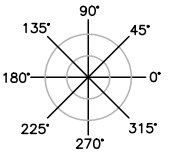
**@** *x* **,** *y*

## 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 .

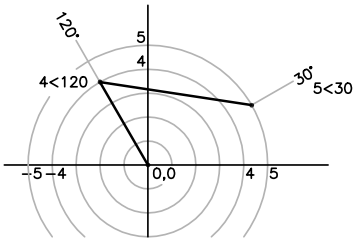



Absolute polar coordinates are measured from the UCS origin (0,0), which is the intersection of the *X* and *Y* axes. Use absolute polar coordinates when you know the precise distance and angle coordinates of the point.

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 335.

The following example shows two lines drawn with absolute polar coordinates using the default angle direction setting. Enter the following in the tooltip:

Command: **line**
From point: **#0,0**
To point: **#4<120**
To point: **#5<30**

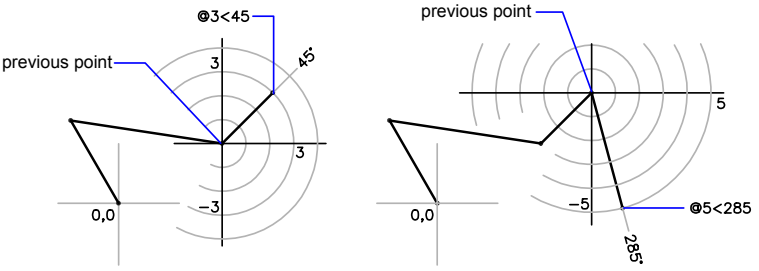



Relative coordinates are based on the last point entered. Use relative coordinates when you know the location of a point in relation to the previous point.

To specify relative coordinates, precede the coordinate values with an @ sign. For example, entering **@1<45** specifies a point at a distance of 1 unit from the last point specified at an angle of 45 degrees from the *X* axis.

The following example shows two lines drawn with relative polar coordinates. In each illustration, the line begins at the location labeled as the previous point.

Command: **line**
From point: **@3<45**
To point: **@5<285**




**To enter absolute polar coordinates (2D)**

- At a prompt for a point, enter coordinates in the tooltip using the following format:

  **# *distance* < *angle***

If Dynamic Input is turned off, enter coordinates on the command line using the following format:

***distance < angle***

**To enter relative polar coordinates (2D)**

- At a prompt for a point, enter coordinates using the following format:

  **@** *distance* **<** *angle*

# 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:

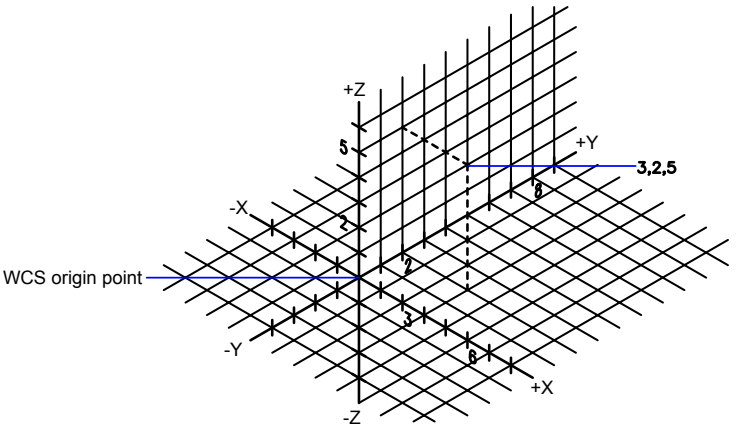
***X,Y,Z***

---

**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 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 on the command line, no prefix is necessary.

### Digitize Coordinates

When you enter coordinates by digitizing, the UCS *Z* value for all coordinates is 0. You can use  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*

## 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 the following syntax:

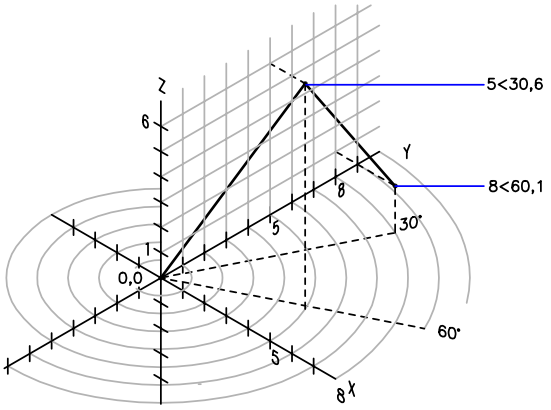
***X* <[ *angle from X axis* ], *Z***

---

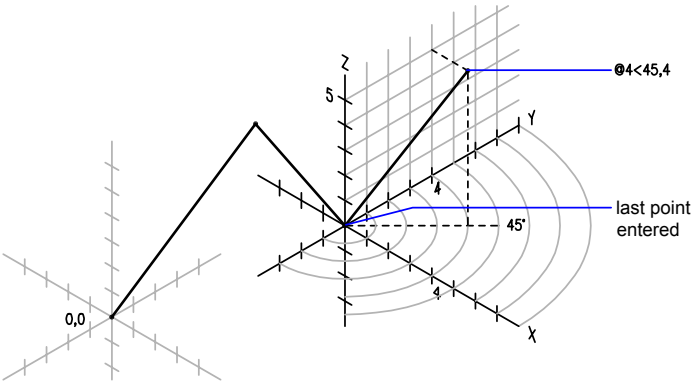
**NOTE** For the following examples, it is assumed that dynamic input is turned off or that the coordinates are entered on the command line. With Dynamic Input, you specify absolute coordinates with the **#** prefix.

---

In the illustration below, 5<60,6 indicates a point 5 units from the origin of the current UCS, 60 degrees from the *X* axis in the *XY* plane, and 6 units along the *Z* axis. 8<30,1 indicates a point 8 units from the origin of the current UCS in the *XY* plane, 30 degrees from the *X* axis in the *XY* plane, and 1 unit along the *Z* axis.




When you need to define a point based on a previous point, you can enter relative cylindrical coordinate values with the @ sign. In the illustration below, @4<45,5 indicates a point 4 units in the *XY* plane from the last point entered, at an angle of 45 degrees from the positive *X* direction, and extending 5 units in the *Z* direction.




**To enter relative cylindrical coordinates**

At a prompt for a point, enter the coordinate values using the following format:

**@** *x* **<** *angle from the X axis* **,** *z*

For example, @4<60,2 represents a location that is 4 units along the *X* axis from the last point measured at 60 degrees from the positive *X* axis and at 2 units in the positive *Z* direction.

## Enter Spherical Coordinates

3D spherical coordinates specify a location by a distance from the origin of the current UCS, an angle from the *X* axis in the *XY* plane, and an angle from the *XY* plane.

Spherical coordinate entry in 3D is similar to polar coordinate entry in 2D. You locate a point by specifying its distance from the origin of the current UCS, its angle from the *X* axis (in the *XY* plane), and its angle from the *XY* plane, each angle preceded by an open angle bracket (<) as in the following format:

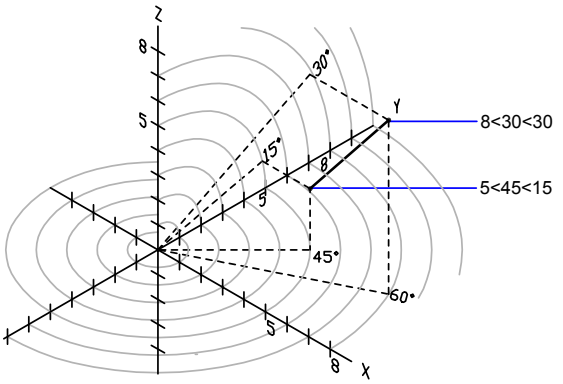
***X* <[ *angle from X axis* ]<[ *angle from XY plane* ]**

---

**NOTE** For the following examples, it is assumed that dynamic input is turned off or that the coordinates are entered on the command line. With Dynamic Input, you specify absolute coordinates with the # prefix.

---

In the following illustration, 8<60<30 indicates a point 8 units from the origin of the current UCS in the *XY* plane, 60 degrees from the *X* axis in the *XY* plane, and 30 degrees up the *Z* axis from the *XY* plane. 5<45<15 indicates a point 5 units from the origin, 45 degrees from the *X* axis in the *XY* plane, and 15 degrees up from the *XY* plane.




When you need to define a point based on a previous point, enter the relative spherical coordinate values by preceding them with the @ sign.

**To enter relative spherical coordinates**

- At a prompt for a point, enter the coordinate values using the following format:

  **@**x<angle from the x axis<angle from the xy plane

For example, @4<60<30 represents a location that is 4 units from the last point measured at 60 degrees from the positive *X* axis in the *XY* plane and at 30 degrees from the *XY* plane.

# Control the User Coordinate System (UCS) in 2D

You can relocate and rotate the user coordinate system for convenient coordinate entry, grid display, grid snap, and Ortho mode.

There are two coordinate systems: a fixed system called the world coordinate system (WCS) and a movable system called the user coordinate system (UCS). In the WCS, the *X* axis is horizontal, the *Y* axis is vertical, and the *Z* axis is perpendicular to the *XY* plane. The origin is where the *X* and *Y* axes intersect (0,0) in the lower-left corner of the drawing. You define a UCS in terms of the WCS. Virtually all coordinate entry uses the current UCS.

Moving the UCS can make it easier to work on particular sections of your drawing. Rotating the UCS helps you specify points in three-dimensional or rotated views. Snap, Grid, and Ortho modes all rotate in line with the new UCS.

You can relocate the user coordinate system using any of the following methods:

- Move the UCS by defining a new origin.
- Align the UCS with an existing object or with the current viewing direction.
- Rotate the current UCS around any of its axes.
- Restore a saved UCS.

Once you have defined a UCS, you can name it and then restore it when you need to use it again. If you no longer need a named UCS, you can delete it. Also, you can restore the UCS to be coincident with the WCS.

**To define a new UCS origin**

1. Click Tools menu ➤ New UCS ➤ Origin.
2. Specify a point for the new origin.

The 0,0,0 coordinate is redefined at the point you specify.

UCS toolbar

**To restore the WCS**

1. Click Tools menu ➤ Named UCS.
2. In the UCS dialog box, Named UCSs tab, select World.
3. Click Set Current.
4. Click OK.

UCS toolbar

**To restore the previous UCS**

1. Click Tools menu ➤ Named UCS.
2. In the UCS dialog box, Named UCSs tab, select Previous.
3. Click Set Current.
4. Click OK.

UCS toolbar

**To save a UCS**

1. Click Tools menu ➤ 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.

UCS toolbar

**To restore a named UCS**

1. Click Tools menu ➤ 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.

UCS toolbar

**To rename a UCS**

1. Click Tools menu ➤ 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.

UCS toolbar

**To delete a UCS**

1. Click Tools menu ➤ 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.

UCS toolbar

# Specify Drawing Planes in 3D (UCS)

Control of the user coordinate system is essential for effective 3D modeling.

## Use World and User Coordinate Systems in 3D

When you work in 3D, there is both a fixed coordinate system and a movable one. The movable user coordinate system is useful for entering coordinates, establishing drawing planes, and setting views.

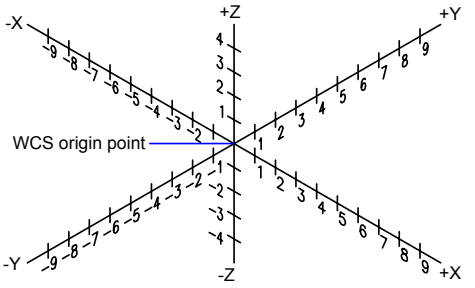
There are two coordinate systems: a fixed system called the world coordinate system (WCS) and a movable system called the user coordinate system (UCS). The UCS is useful for entering coordinates, defining drawing planes, and setting views. Changing the UCS does not change your viewpoint. It changes only the orientation and tilt of the coordinate system.

If you are creating 3D objects, you can relocate the UCS to simplify your work. For example, if you have created a 3D box, you can edit each of its six sides easily by aligning the UCS with each side as you edit it.

You relocate a UCS by choosing the location of the origin point and the orientation of the *XY* plane and the *Z* axis. You can locate and orient a UCS anywhere in 3D space. Only one UCS is current at any given time, and all coordinate input and display is relative to it. If multiple viewports are displayed, they share the current UCS.

However, with the system variable turned on, you can lock a UCS to a viewport, automatically restoring the UCS each time that viewport is made current.

When drawing in 3D, you specify *X*, *Y*, and *Z* coordinate values in either the world coordinate system or the current user coordinate system. The illustration below shows the *X*, *Y*, and *Z* axes of the WCS.

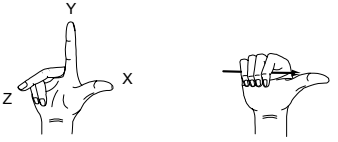



The WCS and the UCS are often coincident — their axes and origin points overlap exactly. No matter how you reorient the UCS, you can always make it coincident with the WCS by using the World option of the UCS command.

### Apply the Right-Hand Rule

Use the right-hand rule to determine the positive axis direction of the *Z* axis when you know the direction of the *X* and *Y* axes in a 3D coordinate system. Place the back of your right hand near the screen and point your thumb in the direction of the positive *X* axis. Extend your index and middle fingers as illustrated, pointing your index finger in the direction of the positive *Y* axis. Your middle finger indicates the direction of the positive *Z* axis. By rotating your hand, you see how the *X*, *Y*, and *Z* axes rotate as you change the UCS.

You can also use the right-hand rule for determining the positive direction of rotation about an axis in 3D space. Point your right thumb in the positive direction of the axis and curl your fingers. Your fingers indicate the positive rotation direction about the axis.




### To enter coordinates relative to the WCS

- Precede coordinate values with an asterisk (*).

Entering @***2,0,0** specifies a point two units in the *X* direction of the last point entered relative to the WCS. Entering @**2,0,0** specifies a point two units in the *X* direction of the last point entered relative to the UCS.

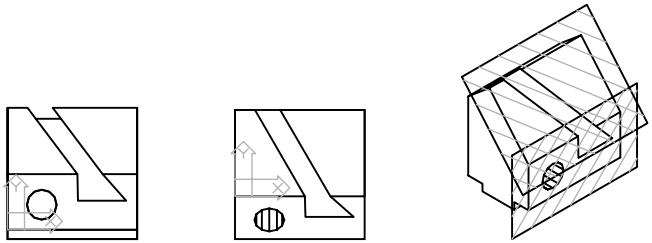
In practice, most coordinates are entered relative to the UCS rather than the WCS.

## Control the User Coordinate System in 3D

Several methods are available for manipulating the user coordinate system in 3D. You can also save and restore user coordinate system orientations.

You define a user coordinate system (UCS) to change the location of the 0,0,0 origin point and the orientation of the *XY* plane and *Z* axis. You can locate and orient a UCS anywhere in 3D space, and you can define, save, and recall as many UCSs as you require. Coordinate input and display are relative to the current UCS.

UCSs are especially useful in 3D. You may find it easier to align the coordinate system with existing geometry than to figure out the exact placement of a 3D point.




If multiple viewports are active, you can assign a different UCS to each viewport. With the  system variable turned on, you can lock a UCS to a viewport, automatically restoring the UCS each time that viewport is made current. With UCSVP turned on, each UCS can have a different origin and orientation for various construction requirements.

### Define the UCS Location

You can define a UCS in the following ways:

- Specify a new origin, new *XY* plane, or new *Z* axis.
- Align the new UCS with an existing object.

- Align the new UCS with the current viewing direction.
- Rotate the current UCS around any of its axes.
- Apply a new *Z*-depth to an existing UCS.
- Apply a UCS by selecting a face.

#### Use UCS Presets

If you do not want to define your own UCS, you can choose from several preset coordinate systems. The images on the Orthographic UCSs tab of the UCS dialog box show the available choices.

If you have already specified a UCS, you can control whether choosing a preset option shifts the UCS relative to the current UCS orientation or relative to the default world coordinate system (WCS). This option has no effect if you restore the WCS, restore the previous UCS, or set the UCS to the current view.

#### Change the Default Elevation

The  command sets the default *Z* value for new objects above or below the *XY* plane of the current UCS. This value is stored in the ELEVATION system variable.

Generally, it is recommended that you leave the elevation set to zero and control the *XY* plane of the current UCS with the  command.

When the UCSVP system variable is set to 1, that is, when different UCS settings are saved in individual viewports, the default elevation for new objects in each viewport is relative to the UCS in that viewport.

#### Change the UCS in Paper Space

You can define a new UCS in paper space just as you can in model space; however, the UCSs in paper space are restricted to 2D manipulation. Although you can enter 3D coordinates in paper space, you cannot use 3D viewing commands such as , , and . The program keeps track of the last 10 coordinate systems created in model space and the last 10 in paper space.

#### Save and Restore UCS Locations by Name

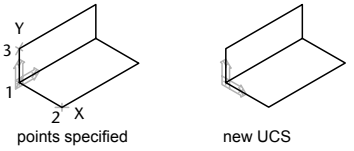
If you plan to work extensively in 3D, you can save named UCS locations, each having a different origin and orientation, for various construction requirements. You can relocate, save, and recall as many UCS orientations as you require.

### To shift the XY plane

1. Click Tools menu ➤ New UCS ➤ 3 Point.
2. Specify an origin point for the new UCS (1).

   For example, in a large drawing, you might specify an origin point near the area in which you want to work.
3. Specify a point to indicate the horizontal orientation of the new UCS (2). This point should be on the positive portion of the new *X* axis.
4. Specify a point to indicate the vertical orientation of the new UCS (3). This point should be on the positive portion of the new *Y* axis.

   The UCS, including grid, shifts to represent the *X* and *Y* axes you have specified.




UCS toolbar

### To select a UCS preset

1. Click Tools menu ➤ Orthographic UCS ➤ Preset.
2. In the UCS dialog box, Orthographics UCSs tab, select a UCS orientation from the list.
3. Click Set Current.
4. Click OK.

   The UCS changes to the selected option.

UCS II toolbar

**To restore the location and orientation of the previous UCS**

■ On the command line, enter **ucs**. Then enter **p** (Previous).

The previous UCS is restored.

UCS toolbar

# Assign User Coordinate System Orientations to Viewports

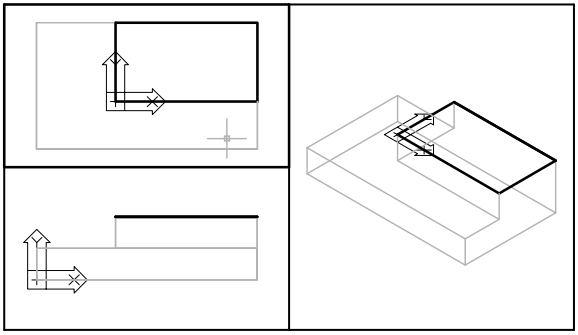
To facilitate editing objects in different views, you can define a different user coordinate system orientation for each view.

Multiple viewports provide different views of your model. For example, you might set up viewports that display top, front, right side, and isometric views. To facilitate editing objects in different views, you can define a different UCS for each view. Each time you make a viewport current, you can begin drawing using the same UCS you used the last time that viewport was current.

The UCS in each viewport is controlled by the  system variable. When UCSVP is set to 1 in a viewport, the UCS last used in that viewport is saved with the viewport and is restored when the viewport is made current again. When UCSVP is set to 0 in a viewport, its UCS is always the same as the UCS in the current viewport.

For example, you might set up three viewports: a top view, front view, and isometric view. If you set the UCSVP system variable to 0 in the isometric viewport, you can use the Top UCS in both the top viewport and the isometric viewport. When you make the top viewport current, the isometric viewport's UCS reflects the UCS top viewport. Likewise, making the front viewport current switches the isometric viewport's UCS to match that of the front viewport.

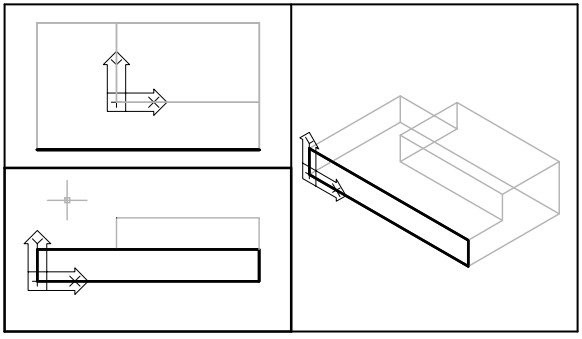
The example is illustrated in the following figures. The first figure shows the isometric viewport reflecting the UCS of the upper-left, or top, viewport, which is current.



top viewport is current

UCSVP = 0; UCS icon reflects UCS of current viewport

The second figure shows the change that occurs when the lower-left, or front, viewport is made current. The UCS in the isometric viewport is updated to reflect the UCS of the front viewport.



front viewport is current

UCSVP = 0; UCS icon reflects UCS of current viewport

In previous releases, the UCS was a global setting for all viewports in either model or paper space. If you want to restore the behavior of previous releases, you can set the value of the UCSVP system variable to 0 in all active viewports.

**To apply the current UCS to other viewports**

1 Make sure the UCS that you want to apply to another viewport is current.

2 Click Tools menu ➤ New UCS ➤ Apply.

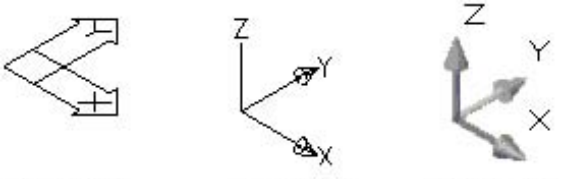
3 At the prompt, click in a viewport to apply the current UCS to that viewport, or enter **all** to apply the current UCS setting to all active viewports.

# Control the Display of the User Coordinate System Icon

To help visualize the current orientation of the user coordinate system, you can display the user coordinate system icon. Several versions of this icon are available, and you can change its size, location, and color.

To indicate the location and orientation of the UCS, the UCS icon is displayed either at the UCS origin point or in the lower-left corner of the current viewport.

You can choose one of three styles of icon to represent the UCS.



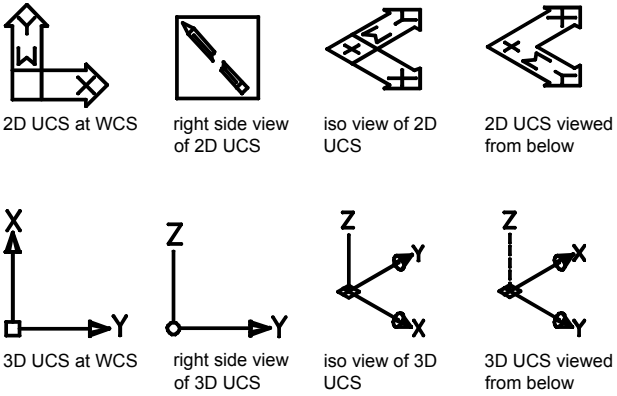

2D UCS icon 3D UCS icon shaded UCS icon

Use the  command to choose between displaying the 2D or the 3D UCS icon. Use  to display the shaded UCS icon. To indicate the origin and orientation of the UCS, you can display the UCS icon at the UCS origin point using the UCSICON command.

If the icon is displayed at the origin of the current UCS, a cross (+) appears in the icon. If the icon is displayed in the lower-left corner of the viewport, no cross appears in the icon.

If you have multiple viewports, each viewport displays its own UCS icon.

The UCS icon is displayed in various ways to help you visualize the orientation of the drawing plane. The following figure shows some of the possible icon displays.




You can use the UCSICON command to switch between the 2D UCS icon and the 3D UCS icon. You can also use the command to change the size, color, arrowhead type, and icon line width of the 3D UCS icon.

The UCS broken pencil icon replaces the 2D UCS icon when the viewing direction is in a plane parallel to the UCS *XY* plane. The broken pencil icon indicates that the edge of the *XY* plane is almost perpendicular to your viewing direction. This icon warns you not to use your pointing device to specify coordinates.

broken pencil icon

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

1. Click View menu ➤ Display ➤ UCS Icon ➤ Select On.

   The check mark indicates whether the icon is on or off.

#### To display the UCS icon at the UCS origin

1. Select View menu ➤ Display ➤ UCS Icon ➤ 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 menu ➤ Display ➤ UCS Icon ➤ Properties.
2. In the UCS Icon dialog box, change the settings.
3. Click OK.

# 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.

The actions required to complete a command or to use grips are similar to those for the command line. 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. Alternatively, you can undock the command window and use Auto-hide to roll open or roll up the window.

#### Turn Dynamic Input On and Off

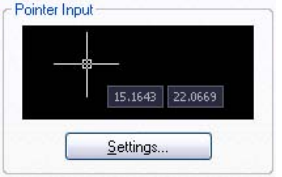
Click Dyn on the status bar to turn Dynamic Input on and off. You can turn it off temporarily by holding down the F12 key. Dynamic Input has three components: pointer input, dimensional input, and dynamic prompts. Right-click Dyn and click Settings to control what is displayed by each component when Dynamic Input is on.

**NOTE** Dynamic Input is not supported for perspective views.

### 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.

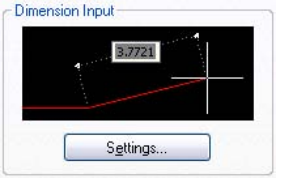
The default for second and subsequent points is relative polar coordinates (relative Cartesian for ). There is no need to type the at sign (@). If you want to use absolute coordinates, use the pound sign (#) prefix. For example, to move an object to the origin, for the second point prompt, enter **#0,0**.




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 , , , , and .




**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.

When you use grips to edit an object, the dimensional input tooltips can display the following information:

Original length

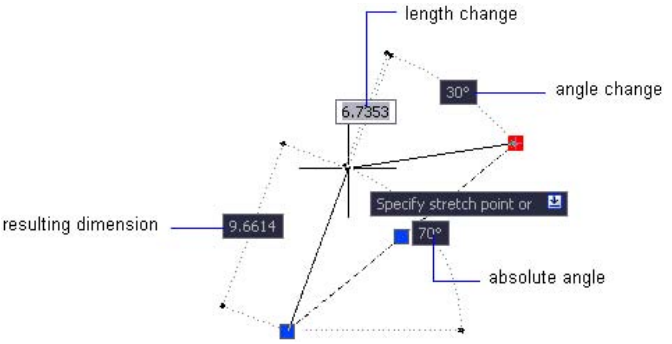
A length that updates as you move the grip

The change in the length

Angle

The change in the angle as you move the grip

The radius of an arc

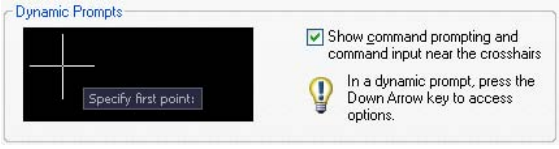



Use the dimensional input settings to display only the information you want to see.

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  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  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 Dynamic Input (the Dyn 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.

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**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.

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**To correct typing errors in dynamic input tooltips**

- When a dynamic input tooltip displays the red error outline, 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.
- Press the UP ARROW key to access recent coordinates, or right-click and click Recent Input to access the coordinates from a shortcut menu.
- 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  system variable.
- To enter relative coordinates when absolute coordinates are displayed, enter @ to temporarily override the  system variable.
- To enter absolute world coordinate system (WCS) coordinates, enter *.

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**NOTE** During pointer input, you can use the shortcut menu to access the # and * prefixes.

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### To choose options in dynamic prompt tooltips

1. Start a command.

   Tooltips near the crosshairs display the coordinates of the cursor location and a prompt.
2. If a prompt is displayed, press the DOWN ARROW key to display the options.
3. Press DOWN ARROW or UP ARROW to place a dot next to the option you want. Press ENTER.

   Press ESC to undo the most recent action.
4. Specify points and enter options to complete the command.

### To change the color, size, or transparency of tooltips

1. Right-click Dyn on the status bar. Click Settings.
2. In the Drafting Settings dialog box, Dynamic input tab, click Drafting Tooltip Appearance.
3. In the Tooltip Appearance dialog box, under Color, click Model Color or Layout Color to display the "Select Color Dialog Box", where you can specify a color for tooltips in the space you selected.
4. Under Size, move the slider to the right to make tooltips larger or to the left to make them smaller. The default value, 0, is in the middle.
5. Under Transparency, move the slider. The lower the setting, the more transparent the tooltip. A value of 100 sets the tooltip to opaque.
6. Under Apply To, choose an option:

   - **Override OS Settings for All Drafting Tooltips**. Applies the settings to all tooltips, overriding the settings in the operating system.
   - **Use Settings Only for Dynamic Input Tooltips**. Applies the settings only to the drafting tooltips used in Dynamic Input.
7. Click OK.

### To turn on or turn off Dynamic Input

- On the status bar, click Dyn, or press F12 .

To turn off Dynamic Input temporarily, hold down the F12 key while you work.

**NOTE** The F12 temporary override key does not turn on Dynamic Input.

### To change pointer input settings

1. Right-click Dyn on the status bar. Click Settings.
2. In the Drafting Settings dialog box, Dynamic input tab, under Pointer Input, click Settings.
3. In the Pointer Input Settings dialog box, select polar or Cartesian format as the default.
4. Select relative or absolute coordinate format as the default.
5. Under Visibility, select one of the following options:
   - **As Soon As I Type Coordinate Data**. When pointer input is turned on, displays tooltips only when you start to enter coordinate data.
   - **When a Command Asks for a Point**. When pointer input is turned on, displays tooltips whenever a command prompts for a point.
   - **Always—Even When Not in a Command**. Always displays tooltips when pointer input is turned o.
6. Click OK to close each dialog box.

   DSETTINGS

### To change dimensional input settings

1. Right-click Dyn on the status bar. Click Settings.
2. In the Drafting Settings dialog box, Dynamic input tab, under Dimension Input, click Settings.
3. In the Dimension Input Settings dialog box, select Polar or Cartesian format as the default.
4. Under Visibility, select one of the following options:
   - **Show Only 1 Dimension Input Field at a Time**. Displays only the distance dimensional input tooltip when you are using grip editing to stretch an object.
   - **Show 2 Dimension Input Fields at a Time**. Displays the distance and angle dimensional input tooltips when you are using grip editing to stretch an object.

- **Show the Following Dimension Input Fields Simultaneously**. Displays the selected dimensional input tooltips when you are using grip editing to stretch an object. Select one or more of the check boxes.

5 Click OK to close each dialog box.

DSETTINGS

### To display prompts in tooltips

1 Right-click Dyn on the status bar. Click Settings.

2 In the Drafting Settings dialog box, Dynamic input tab, under Dynamic Prompts, check Show Command Prompting and Command Input Near the Crosshairs.

3 Click OK.

DSETTINGS

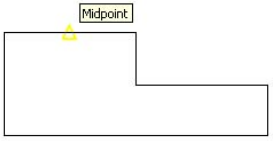
# Snap to Locations on Objects (Object Snaps)

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 .

### 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 on the command line

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 OSNAPHATCH system variable to 0.

---

### Use Object Snaps in 3D

By default, the *Z*-value of an object snap location is determined by the object's location in space. However, if you work with object snaps on the plan view of a building or the top view of a part, a constant *Z*-value is more useful.

If you turn on the  system variable, all object snaps are projected onto the *XY* plane of the current UCS or, if  is set to a non-zero value, onto a plane parallel to *XY* plane at the specified elevation.

---

**NOTE** When you draw or modify objects, make sure that you know whether OSNAPZ is on or off. There is no visual reminder, and you can get unexpected results.

---

### To snap to a geometric point on an object

1. At the prompt for a point, hold down SHIFT and right-click in the drawing area. Select the object snap you want to use.
2. Move your cursor over the desired object snap location.

   If AutoSnap is on, your cursor automatically locks onto the snap location you selected, and a marker and tooltip indicate the object snap point.
3. Select an object.

   The cursor snaps to the eligible location closest to your selection.

### To set running object snaps

1. Click Tools menu ➤ Drafting Settings.
2. In the Drafting Settings dialog box, Object Snap tab, select the object snaps you want to use.
3. Click OK.

Object Snap toolbar

Hold down SHIFT and right-click to display the object snap menu. Click Osnap Settings.

### To turn on and turn off running object snaps as you work

■ On the status bar, click Osnap, or press F3.

If running object snaps have been set, the settings are turned on or off.

To turn running object snaps on and off temporarily, hold down the F3 key while you work.

### To set object snaps to ignore hatch objects

1. At the Command prompt, enter **osnaphatch**.
2. Enter **0**.

# 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.

Object Snap toolbar

# Override Object Snaps

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.

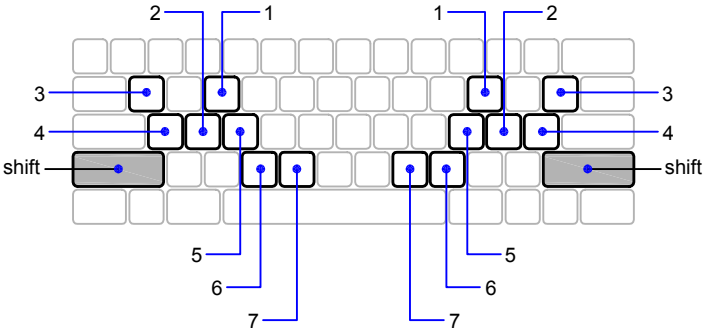
For example, if you have set running object snaps but you want to turn them off for one point, you can hold down F3. When you release this override key, running object snaps are restored.

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.

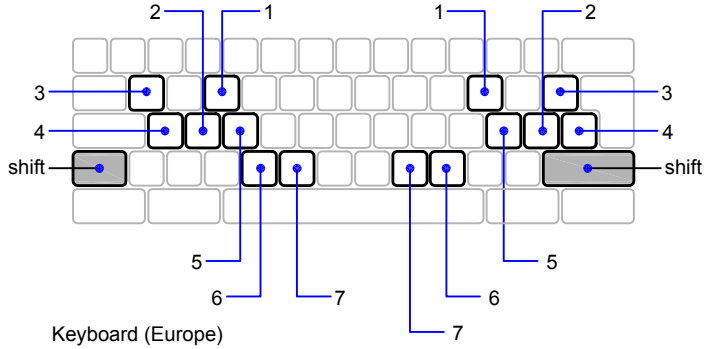
The keys in the following illustration are the default keys, but you can change key assignments and add your own as needed.

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 | | Object snap override: Center |
| 3 | Toggles object snap tracking | 7 | Object snap override: Midpoint |
| 4 | Toggles object snap mode (OSNAP) | | |




Keyboard (U.S. and Japan)




Keyboard (Europe)

Temporary override keys are also available for the other drawing aids that you set in the "Drafting Settings Dialog Box".

**See also:**

"Adjust Grid and Grid Snap"
"Use Orthogonal Locking (Ortho Mode)"
"Use Polar Tracking and PolarSnap"
"Use Dynamic Input"
**Add Shortcut Keys and Temporary Override Keys 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.

# 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 your drawing speed and efficiency, you can display and snap to a rectangular pattern of grid points. You can also control its spacing, angle, and alignment.

The grid is a rectangular pattern of dots 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. If you zoom in or out of your drawing, you may need to adjust grid spacing to be more appropriate for the new magnification.

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.

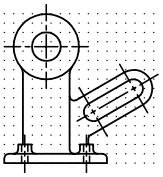
### Change Grid and Snap Spacing

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.

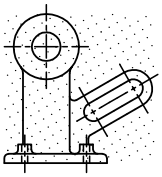
Snap spacing does not have to match grid spacing. For example, you might set a wide grid spacing to be used as a reference but maintain a closer snap spacing for accuracy in specifying points.

### Change the Snap Angle and Base Point

If you need to draw along a specific alignment or angle, you can change the snap angle. This rotation realigns the crosshairs on the screen to match the new angle. In the following example, the snap angle is adjusted to match the angle of the anchor bracket.



default snap angle - 0 degrees



rotated snap angle - 30 degrees

When you set the snap angle, the grid rotation also changes.

The snap base point is the offset from a specified *X,Y* value from which snap points are measured. The default snap base point is 0,0, but it can be changed to any X,Y coordinate value to adjust snap points.

For example, if *X* and *Y* spacing are set to 0.5, and *X* and *Y* base points are set to 0,0, the snap points along the *X* and *Y* axes are 0, 0.50, 1.0, and so on. Changing the *X* snap base point to 0.1 and the *Y* snap base point to 0.2 causes the snap points along the *X* axis to shift to 0.1, 0.60, 1.1, and so on, and the snap points along the *Y* axis to shift to 0.2, 0.7, 1.2, and so on. Shifting the snap base point may be necessary to align a hatch pattern.

**See also:**

"**Set Isometric Grid and Snap**"
"**Override Object Snaps**"

### To display a grid and set grid spacing

1. Click Tools menu ➤ Drafting Settings.
2. In the Drafting Settings dialog box, Snap & Grid tab, select Grid On to display the grid.
3. Under Snap Type and Style, 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.

   Although the grid dots do not necessarily correspond to the current snap interval, they do reflect the current snap angle and isometric snap and grid settings.

### To turn on Snap mode and set snap spacing

1. Click Tools menu ➤ Drafting Settings.
2. In the Drafting Settings dialog box, Snap & Grid tab, select Snap On.
3. Under Snap Type and Style, make sure Grid Snap and Rectangular Snap are selected.
4. In Snap Type & Style, select Grid Snap.
5. In the Grid *X* Spacing box, enter the horizontal snap spacing value in units.
6. To specify the same vertical snap spacing, press ENTER. Otherwise, enter a new distance in the Grid *Y* Spacing box.
7. Click OK.

### To temporarily override Snap mode

■ Hold down F9 while you work.

When you release the key, Snap mode is restored.

### To rotate the snap angle and change its base point

1. Click Tools menu ➤ Drafting Settings.
2. In the Drafting Settings dialog box, Snap & Grid tab, under Snap, enter a rotation angle in the Angle box.
3. If you want to set the base point in order to align snap locations, enter the *X* and *Y* coordinate values in the X Base and Y Base boxes.
4. Click OK.

## 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 orthogonal alignment depends on the current snap angle, UCS, or isometric grid and snap setting.

Use Ortho mode to draw more quickly. For example, create a series of perpendicular lines by turning on Ortho mode before you start.

You can turn Ortho on and off at any time during drawing and editing. Ortho is ignored when you enter coordinates on the command line, use perspective views, 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 350.

**NOTE** Ortho mode and polar tracking cannot be on at the same time. Turning on Ortho turns off polar tracking.

**See also:**

"Override Object Snaps"

**To turn on or turn off Ortho mode**

- On the status bar, click Ortho.

  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.

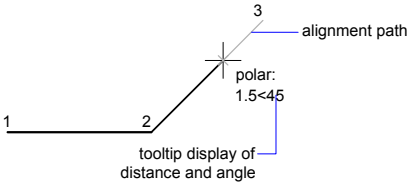
  **NOTE** Turning on Ortho automatically turns off polar tracking.

# 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. You can use PolarSnap™ to snap to specified distances along the alignment path.

For example, in the following illustration you draw a two-unit line from point 1 to point 2, and then draw a two-unit line to point 3 at a 45-degree angle to the line. If you turn on the 45-degree polar angle increment, an alignment path and tooltip are displayed when your cursor crosses the 0 or 45-degree angle. The alignment path and tooltip disappear when you move the cursor away from the angle.




As you move your cursor, alignment paths and tooltips are displayed when you move the cursor near polar angles. The default angle measurement is 90 degrees. Use the alignment path and tooltip to draw your object. You can use polar tracking with Intersection and Apparent Intersection object snaps to find where a polar alignment path intersects another object.

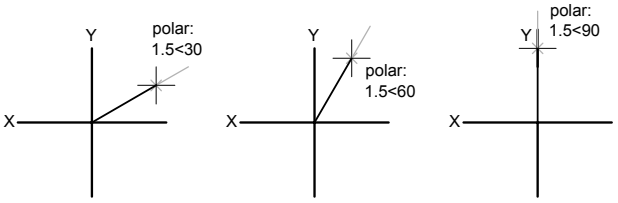
---

**NOTE** Ortho mode and polar tracking cannot be on at the same time. Turning on polar tracking turns off Ortho mode. Similarly, PolarSnap and grid snap cannot be on at the same time. Turning on PolarSnap turns off grid snap.

---

### Specify Polar Angles (Polar Tracking)

You can use polar tracking to track along polar angle increments of 90, 60, 45, 30, 22.5, 18, 15, 10, and 5 degrees, or you can specify other angles. The following illustration shows the alignment paths displayed as you move your cursor 90 degrees with the polar angle increment set to 30 degrees.




The orientation of 0 depends on the angle you set in the Drawing Units dialog box (). The direction of snap (clockwise or counterclockwise) depends on the units direction you specify when setting units of measurement.

You can turn polar tracking on and off temporarily by using an override key. The direct distance entry method is not available while you are using the temporary override key for polar tracking.

### Specify Polar Distances (PolarSnap)

PolarSnap restricts cursor movement to increments of a polar distance you specify. For example, if you specify a length of 4 units, the cursor snaps from the first point specified to lengths of 0, 4, 8, 12, 16, and so on. As you move your cursor, a tooltip indicates the nearest PolarSnap increment. To restrict point entry to polar distances, both polar tracking and Snap mode (set to PolarSnap) must be on. You can turn off all snapping and tracking temporarily by using an override key.

**See also:**

"Override Object Snaps"

#### To turn on and turn off polar tracking

■ Press F10, or click Polar 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 & Style, 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.

# 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  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.

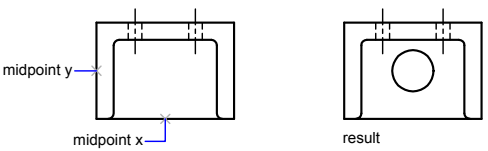
Coordinate filters specify a new coordinate location by using the *X* value from one location, the *Y* value of a second location, and, for 3D coordinates, the *Z* value of a third location. When used with object snaps, coordinate filters extract coordinate values from an existing object.

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 on the command line, 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.



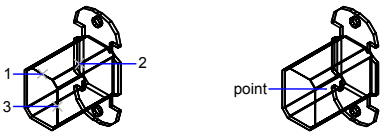

Here is the command line sequence:

Command: **circle**
Specify center point for circle or [3P/2P/Ttr (tangent tangent radius)]: .**x**
of: **mid**
of: *Select the horizontal line on the lower edge of the holding plate*
of: (need YZ): **mid**
of: *Select the vertical line on the left side of the holding plate*
of: Diameter/<Radius> *Specify the radius of the hole*
Coordinate filters work only when the program prompts you for a point. If you try to use a coordinate filter at the Command prompt, you see an error message.

### Example: Use of Coordinate Filters in 3D

This example shows how to use coordinate filters to create a point object at the center (centroid) of a 3D object. Hidden lines have been removed for clarity. The *X* value of the new point is extracted from the first location specified, the *Y* value from the second location, and the *Z* value from the third. The three values are combined to form the coordinate values of the new point.

Command: **point**
Point: **.x**
of mid
of select object (*1*)
(need YZ): **.y**
**of mid**
of select object (*2*)
(need Z): **mid**
of select object (*3*)




### To use coordinate filters to specify a point in 2D

1. At the prompt for a point, enter a coordinate filter (**.x** or **.y**).

   For example, enter **.x** to specify the *X* value first.

2. To extract the first coordinate value, specify a point.

   For example, if you entered **.x** in step 1, the *X* value is extracted from this point.

3. To extract the next coordinate value, specify a different point.

   The new point location combines the coordinate values extracted from the points you specified in steps 2 and 3.

---

**NOTE** Instead of specifying a point in steps 2 or 3, you can enter a numeric value.

---

### To use coordinate filters to specify a point in 3D

1. At the prompt for a point, enter a coordinate filter (**.x**, **.y**, **.z**, **.xy**, **.xz**, or **.yz**).

For example, enter **.x** to specify the *X* value first.

2 To extract the specified coordinate value(s), specify a point.

For example, if you entered **.x** in step 1, the *X* value is extracted from this point.

3 At the prompt for the remaining coordinates, do one of the following:

- Extract the remaining coordinate values by specifying a point.
- Enter another coordinate filter and return to step 2.

For example, if you entered **.x** in step 1, specify a second point to extract the *Y* and *Z* coordinates simultaneously, or enter **.y** or **.z** to specify *Y* and *Z* values separately.

The new point location combines the coordinate values extracted from the points specified in steps 2 and 3.

**NOTE** Instead of specifying a point in steps 2 or 3, you can enter a numeric value.

# 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.

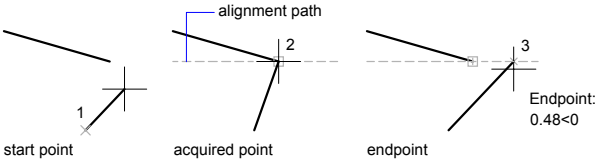
You can toggle AutoTrack on and off with the Polar and Otrack buttons on the status bar. Use temporary override keys to turn object snap tracking on and off or to turn off all snapping and tracking. See the keyboard illustration in "Override Object Snaps" on page 345.

Object snap tracking works in conjunction with object snaps. You must set an object snap before you can track from an object's snap point.

### Object Snap Tracking

Use object snap tracking to track along alignment paths that are based on object snap points. Acquired points display a small plus sign (+), and you can acquire up to seven tracking points at a time. After you acquire a point, horizontal, vertical, or polar alignment paths relative to the point are displayed as you move the cursor over their drawing paths. For example, you can select a point along a path based on an object endpoint or midpoint or an intersection between objects.

In the following illustration, the Endpoint object snap is on. You start a line by clicking its start point (1), move the cursor over another line's endpoint (2) to acquire it, and then move the cursor along the horizontal alignment path to locate the endpoint you want for the line you are drawing (3).




### Change Object Snap Tracking Settings

By default, object snap tracking is set to orthogonal. Alignment paths are displayed at 0, 90, 180, and 270 degrees from acquired object points. However, you can use polar tracking angles instead.

For object snap tracking, object points are automatically acquired. However, you can choose to acquire points only when you pressSHIFT.

### 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 Otrack on the status bar.

#### 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.

# 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 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 350
"Lock an Angle for One Point (Angle)" on page 353

**To draw a line using direct distance entry**

1. Start the LINE command and specify the first point.
2. 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 on the command line.

   The line is drawn at the length and angle you specified.

## 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**.
2. If you want to offset from a location on an existing object, specify an object snap method. Then select the object.
3. Enter a relative coordinate.

   Press SHIFT and right-click to display the object snap menu.

Object Snap toolbar

# 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 ()
- Specify the number of equal segments ()

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 .

**See also:**

"Create and Use Blocks (Symbols)" on page 451

## Specify Measured Intervals on Objects

You can mark off equal lengths from one end of a selected object.

You can use  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 , or choose Point Style from the Format menu. The  system variable also controls the appearance of point markers. For example, you can change the value to make points appear as crosses.  controls the size of point objects.

### To insert points at measured intervals on an object

1. Click Draw menu ➤ Point ➤ 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 Draw menu ➤ Point ➤ 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.

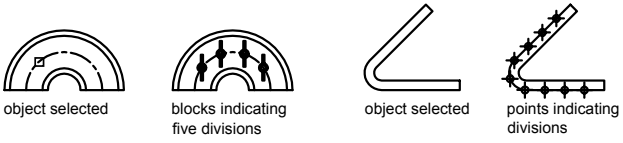
Blocks are inserted on the object at the specified intervals.

,

## 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 , or choose Point Style from the Format menu. The  system variable also controls the appearance of point markers. For example, you can change the value to make points appear as crosses.  controls the size of point objects.

### To insert points to mark equal segments

1 Click Draw menu ➤ Point ➤ 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 Draw menu ➤ Point ➤ 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.

,

# Extract Geometric Information from Objects

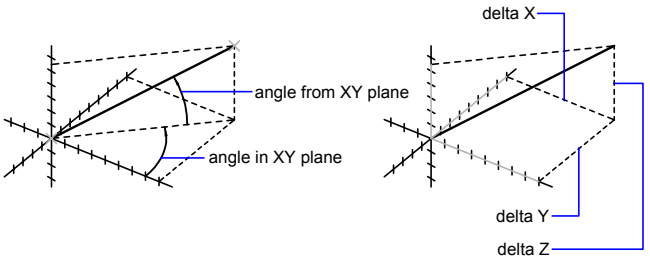
The inquiry and calculation commands can provide information about objects in your drawing and do useful calculations.

## Obtain Distances, Angles, and Point Locations

You can retrieve information about the relation between two specified points: for example, the distance between them or their angle in the *XY* plane.

To determine the relation between two 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  command lists the *X*, *Y*, and *Z* coordinate values of a specified point.

**See also:**

"Overview of Coordinate Entry" on page 312

**To calculate a distance and angle**

1. Click Tools menu ➤ Inquiry ➤ Distance.
2. Specify the first and second points of the distance you want to calculate.

   A brief report is displayed on the command line.

Inquiry toolbar

# Obtain Area Information

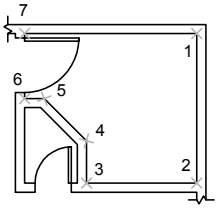
You can obtain the area and perimeter defined by selected objects or a sequence of points.

You can calculate and display the area and perimeter of a sequence of points or of any of several types of objects. If you need to calculate the combined area of more than one object, 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.

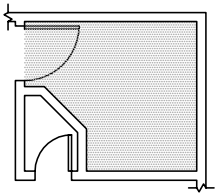
Total area and perimeter are saved in the  and  system variables.

### Calculate a Defined 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.



points specified

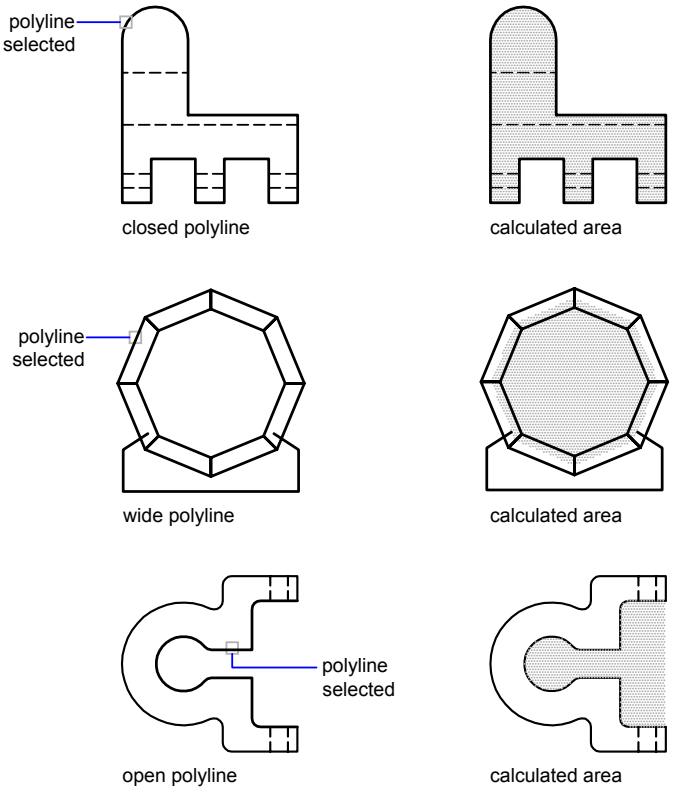

arbitrary closed region

### 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 displayed information differs according to the type of object you select:

- **Circles.** Area and circumference are displayed.
- **Ellipses, closed polylines, polygons, planar closed spline curves, and regions.** Area and perimeter are displayed. 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 are displayed. The area is computed as though a straight line connected the start point and endpoint.
- **AutoCAD 3D solids.** Total 3D area for the object is displayed.

### Example: How Various Areas Are Calculated




### Calculate Combined Areas

You can measure more than one area, either 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

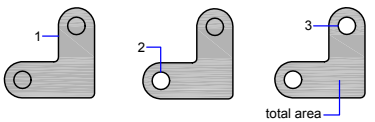
You can subtract one or more areas from a combined area that you have already calculated. In the example, the area of the floor plan is first calculated, and then a room is subtracted.

### Example: Subtraction of Areas from a Calculation

In the following example, the closed polyline represents a metal plate with two large holes. You first calculate the area of the polyline and then subtract each hole. Area and perimeter or circumference of each object are displayed, with a running total after each step.

Here is the command line sequence:

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  to convert the plate and the holes to regions, subtract the holes, and then use the Properties palette or the  command to find the area of the plate.

**See also:**

"Create and Combine Areas (Regions)" on page 421
"Overview of Object Properties" on page 264

### To calculate an area you define

1. Click Tools menu ➤ Inquiry ➤ Area.
2. Specify points in a sequence that defines the perimeter of the area to be measured. Then press ENTER.

The first and last points are connected to form a closed area and the area and perimeter measurements are displayed using the settings specified with UNITS.

Inquiry toolbar

**To calculate the area of an object**

1. Click Tools menu ➤ Inquiry ➤ Area.
2. On the command line, enter **o** (Object).
3. Select an object.

   The area and perimeter of the selected object are displayed.

Inquiry toolbar

**To add areas as you calculate**

1. Click Tools menu ➤ Inquiry ➤ Area.
2. Enter **a** (Add).
3. Use one of the following methods:
   - Specify points to define the area you want to add and press ENTER.
   - Enter **o** (Object) and select the objects you want to add.

     You see the measurements of each new area and a running total of all areas.
4. Press ENTER twice to end the command.

Inquiry toolbar

**To subtract areas from a calculation**

1. While a combined area is displayed, enter **s** (Subtract).

2 Use one of the following methods:

- Specify points to define the area you want to subtract and press ENTER.
- Enter **o** (Object) and select the objects you want to subtract.

The running total is updated as you define new areas.

3 Press ENTER to end the command.

Inquiry toolbar

# 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 line calculator.

# Use the QuickCalc Calculator

With the QuickCalc calculator, an interface that looks and functions like a hand-held calculator, you can perform mathematical, scientific, and geometric calculations, convert units of measurement, manipulate the properties of objects, and evaluate expressions.

## Overview of QuickCalc

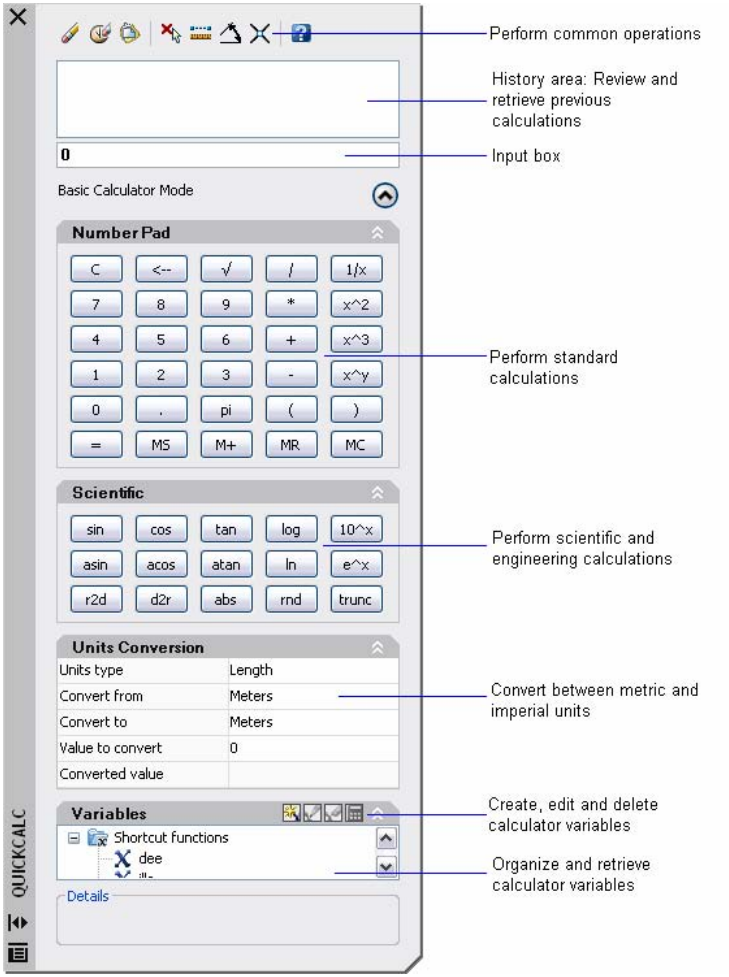
The QuickCalc calculator includes basic features similar to most standard mathematical calculators. In addition, QuickCalc has features specific to AutoCAD such as geometric functions, a Units Conversion area, and a Variables area.

Unlike most calculators, QuickCalc is an expression builder. For greater flexibility, it does not immediately calculate an answer when you click a function. Instead, you compose an expression that you can easily edit and, when you are finished, you click the equal sign (=) or press ENTER. Later, you can retrieve the expression from the History area, modify it, and recalculate the results.

With QuickCalc, you can

- Perform mathematical and trigonometric calculations
- Access and review previously entered calculations for re-evaluation

- Use the calculator with the Properties palette to modify object properties
- Convert units of measurement
- Perform geometric calculations related to specific objects
- Copy and paste values and expressions to and from the Properties palette and the command line
- 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 80.

### To clear the Input box

Do one of the following:

- On the QuickCalc toolbar, click the Clear button.

■ On the QuickCalc Number Pad, click the Clear button.

Any current value or expression in the Input box is cleared and the value is reset to 0.

### To use basic math functions in QuickCalc

1. On the Number Pad, click a number.
2. Click an operator (+, -, *, /) button. Then enter the next number, and so on.
3. Click the equal (=) sign.

   The result is displayed in the Input box. The expression and result are also displayed in the History area.

### To use scientific functions in QuickCalc

1. On the Number Pad, enter a value.
2. In the Scientific area, click a function.
3. On the Number Pad, click the equal (=) sign.
1. The result is displayed in the Input box.

### To get the absolute value of a number using QuickCalc

1. With the current value displayed, in the Scientific area click the abs button.
2. On the Number Pad, click the equal (=) sign.
1. The absolute value is displayed in the Input box.

### To round a number to the nearest integer using QuickCalc

1. With the current value displayed, in the Scientific area click the rnd button.
2. On the Number Pad, click the equal (=) sign.
1. The number, rounded to the nearest integer, is displayed in the Input box.

**To display only the integer portion of a number with QuickCalc**

1. With the current value displayed, in the Scientific area click the trunc button.
2. On the Number Pad, click the equal (=) sign.
1. The integer portion of the number is displayed in the Input box without the decimal portion.

**To store a value in the QuickCalc memory**

- With the current value displayed, click MS.
- The previous value is overwritten and the new value is stored in memory.

**To add the current value to the value stored in the QuickCalc memory**

- With the current value displayed, click M+.
- The sum of the new value and the value already in memory are displayed.

**To restore the value stored north QuickCalc memory**

- With the current value displayed, click MR.
- The displayed value or expression is appended and the value stored in memory is displayed.

**To clear the value stored in the QuickCalc memory**

- Click MC.
- The value currently stored in memory is cleared.

## Access QuickCalc and Understand Its Behavior

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 line
- 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:

- On the Tools menu, click QuickCalc
- On the Standard toolbar, click the QuickCalc button
- In the drawing editor (with no command active), right-click, and then click QuickCalc
- On the command line, enter **quickcalc** or **qc**, and then press ENTER

### 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.
- Enter **'quickcalc** or **'qc** on the command line.

Calculations that you transfer to the command affect the drawing. For example, if you are drawing a line with the LINE command and you pass an expression to the command line 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 line. To use the value, position the crosshairs to determine the direction and then press ENTER.

---

### Use QuickCalc Transparently from the Properties Palette

- 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.

**To use the QuickCalc calculator**

Do *one* of the following:

- On the command line, enter **quickcalc** or **qc** .
- Right-click the drawing area to display a shortcut menu. Click QuickCalc.
- On the Tools menu, click QuickCalc.
- On the Standard toolbar, click QuickCalc.

**To use the QuickCalc calculator within a command**

Do *one* of the following:

- On the command line, enter **'quickcalc** or **'qc**.
- Right-click to display a shortcut menu. Click QuickCalc.

**To use the QuickCalc calculator with the Properties palette**

1. Open the Properties palette.
2. Click one or more objects.
3. Click a box with a numeric property.
4. Click the QuickCalc button in the box.

## 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 line 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**

#### To copy and paste a value from the QuickCalc Input box

- Right-click the value in the Input box. Click Copy.

  The current entry is copied to the clipboard.
- Click in the new location and then right-click. Click Paste.

  The value is copied to the new location.

#### To paste a value from the QuickCalc Input box to the command line

1. On the QuickCalc toolbar, click the Paste Value to Command Line button.

   The value in the Input box is pasted into the command line.

**To clear the History area**

■ On the QuickCalc toolbar, click the Clear History button.

Right-click the History area. Click Clear History.

**To reuse a stored value or expression in the QuickCalc History area**

1. Click the C button to clear the Input box if needed.
2. In the History area, double-click a value or expression. The value or expression is displayed in the Input box.

**NOTE** The cursor must be on the value or expression to select it.

**To change the font color of values or expressions in the QuickCalc History area**

1. Right-click in the History area. Click either Value Font Color or Expression Font Color.
2. In the Color dialog box, click a Basic Color or click Define Custom Colors.

   With Define Custom Colors, you can select a custom color and add it to Custom Colors.
3. Click OK.

2. The colors you selected for values and expressions in the History area are displayed.

   Right-click Expression Font Color or Value Font Color. Click Basic Color or Define Custom Colors.

**To copy and paste an expression from the QuickCalc History area**

■ Right-click the expression in the History area. Click Copy.

The current entry is copied to the clipboard.

■ Right-click in the new location. Click Paste.

The expression is copied to the new location.

**To append a value or expression from the QuickCalc History area to the Input box**

■ Double-click the value or expression in the History area.

The value or expression is appended to the Input box.

**NOTE** The cursor must be on the value or expression to select it.

Right-click Append Value to Input Area or Append Expression to Input Area.

**To modify a property in the Properties palette with QuickCalc**

1. Select an object.

   **NOTE** If QuickCalc is displayed on your desktop when you work with the Properties palette, it is temporarily hidden while you use the modal calculator from within the Properties palette.

2. In the Geometry section of the Properties palette, click on the value of a property.

   A small calculator icon is displayed to the right of the value.

   **NOTE** Only properties that are displayed with a white background can be changed.

3. Click the calculator icon.

   QuickCalc opens and displays the current value of the object in the Input box.

4. Perform a calculation on the displayed value and click the equal sign (=) button.

   The new value is displayed in the Input box.

5. Click Apply.

   **NOTE** The Apply button is only available for editable number-based properties.

   The calculator closes and the new value is displayed in the Properties palette. The object is modified in the drawing.

**To obtain X, Y, Z coordinate values for a point using QuickCalc**

**NOTE** The QuickCalc toolbar Get Coordinates button uses the cur function.

1. 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**

**NOTE** The QuickCalc toolbar Distance Between Two Points button uses the dist(p1,p2) function.

1. 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**

**NOTE** The QuickCalc toolbar Angle of Line Defined by Two Points button uses the ang(p1,p2) function.

1. On the QuickCalc toolbar, click the Angle of Line Defined by Two Points button.

   QuickCalc temporarily closes and you are prompted at the command line 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**

**NOTE** The QuickCalc Intersection of Two Lines Defined by Four Points button uses the ill(p1,p2,p3,p4) function.

1. On the QuickCalc toolbar, click the Intersection of Two Lines Defined by Four Points button.

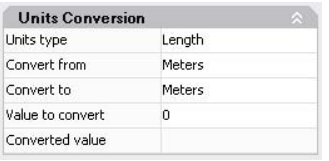
   QuickCalc temporarily closes and you are prompted at the command line to specify four points.
2. Enter the coordinate values for the first point of line one, then the second point of line one. Next, enter the coordinate values for the first point of line two, then the second point of line two.

   QuickCalc opens and appends the value of the evaluated expression to the end of any value or expression already present in the Input box.

# Convert Units of Measurement

In the Units Conversion area of QuickCalc, you can obtain equivalent values for different units of measurement.

Unit conversions are available for length, area, volume, and angular values. Based on which unit type you select, you can then select a list of units to convert from and a list of units to convert to.




The Value to Convert box automatically displays the value from the Input box. You can also enter a different value. The results of the units conversion displays in the Converted Value box. You can paste this result to the Input box by clicking the QuickCalc icon in the Converted Value box.

**NOTE** In the Value to Convert box, enter decimal values without units.

### To convert units of measurement with QuickCalc

1. In the Units Conversion area, select a unit category on the Units Type list.
2. In the Convert From list, select the type of unit you are converting from.
3. In the Convert To list, select the type of unit you are converting to.
4. In the Value to Convert box, enter the value you want to convert. Press ENTER.

1. The converted value is displayed in the Converted Value box.

### To copy a unit conversion result to the Input box of QuickCalc

1. On the Units Conversion title bar, click the Return Conversion Value to Input Area button.

1. 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.

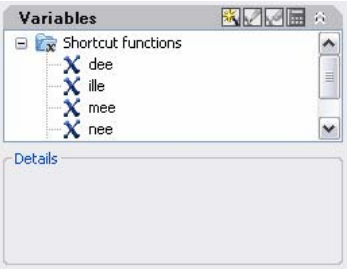
3 On the Number Pad, click the equal (=) sign.

The conversion is displayed in the Input box.

## Create and Use Calculator Variables

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 functions with the Endpoint Snap mode. The following table describes the predefined variables that are available in the Variables area of the calculator.

| Variable | Shortcut For | Description |
|---|---|---|
| dee | dist(end,end) | Distance between two endpoints |
| ille | ill(end,end,end) | Intersection of two lines defined by four endpoints |
| mee | (end+end)/2 | Midpoint between two endpoints |
| nee | nor(end,end) | Unit vector in the XY plane and normal to two endpoints |
| rad | rad | Radius of a selected circle, arc, or polyline arc |
| vee | vee(end,end) | Vector from two endpoints |
| vee1 | vec1(end,end) | Unit vector from two endpoints |

You can easily modify these calculator variables or create your own. For more information, see the  command.

### Organize Variables into Categories

You can organize calculator variables in the Variables area under several categories. This results in a one-level tree structure. The Shortcut Functions category has already been created and contains several functions.

Use the shortcut menu in the Variables area to create, rename, or delete variable categories.

#### To use a predefined variable in a QuickCalc expression

1. In the Variables area, click the variable you want to use.
2. On the QuickCalc Variables title bar, click the Return Variable to Input Area button.

   The variable is displayed in the Input box as part of your expression.

#### To create a new variable in QuickCalc

1. On the QuickCalc Variables title bar, click the New Variable button.
2. In the Variable Definition dialog box, under Variable Type, select Constant or Function.

3 In the Variable Definition dialog box, under Variable Properties Name, enter a name for the variable. Variable names cannot contain spaces or special characters.

**NOTE** When referencing this variable from the Input box, the variable name must begin with the dollar sign ($) to differentiate it from local LISP variables.

4 Under Variable Properties Group With, click New.

5 In the Category Definition dialog box, under Category Properties Name, enter a name for the new category.

6 Under Description, enter a description for the new category. Click OK.

7 In the Variable Definition dialog box, under Value or Expression, enter a value or expression for the new variable.

8 Under Description, enter a description of the new variable. Click OK.

The new variable is now displayed in the Variables area.

### To edit a variable in QuickCalc

1 In the Variables area, click the variable you want to edit.

2 On the QuickCalc Variables title bar, click the Edit Variable button.

3 In the Variable Definition dialog box, make the edits to the variable. Click OK.

### To delete a variable in QuickCalc

1 In the Variables area, click the variable you want to delete.

2 On the QuickCalc Variables title bar, click the Delete button.

### To create a new global constant in the Input box of QuickCalc

1 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.

# Use the Command Line Calculator

By entering an expression in the command line calculator, you can quickly solve a mathematical problem or locate points in your drawing.

The  command runs the 3D calculator utility to evaluate vector expressions (combining points, vectors, and numbers) and real and integer expressions. The calculator performs standard mathematical functions. It also contains a set of specialized functions for calculations involving points, vectors, and AutoCAD geometry. With the CAL command, you can

- Calculate a vector from two points, the length of a vector, a normal vector (perpendicular to the *XY* plane), or a point on a line
- Calculate a distance, radius, or angle
- Specify a point with the pointing device
- Specify the last-specified point or intersection
- Use object snaps as variables in an expression
- Convert points between a UCS and the WCS
- Filter the *X*, *Y*, and *Z* components of a vector
- Rotate a point around an axis

### Evaluating Expressions

CAL evaluates expressions according to standard mathematical rules of precedence.

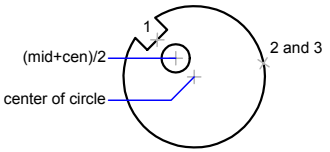
| Mathematical operators in order of precedence | |
|---|---|
| **Operator** | **Operation** |
| ( ) | Groups expressions |
| ^ | Indicates numeric exponent |
| *, / | Multiplies and divides numbers |
| +, – | Adds and subtracts numbers |

### Calculating Points

You can use CAL whenever you need to calculate a point or a number within a command.

For example, you enter **(mid+cen)/2** to specify a point halfway between the midpoint of a line and the center of a circle.

The following example uses CAL as a construction tool. It locates a center point for a new circle, and then calculates one fifth of the radius of an existing circle.




Here is the command line 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)*

No procedures for this topic.

# Draw Geometric Objects

# 19

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 on the command line.

## In this chapter


- Draw Linear Objects
- Draw Curved Objects
- Draw Construction and Reference Geometry
- Create and Combine Areas (Regions)
- Create 3D Objects
- Create Revision Clouds

# Draw Linear Objects

A line, the most basic object, can be one segment or a series of connected segments.

## Draw Lines

In a simple line with connected segments, each segment is a separate line object.

With , you can create a series of contiguous line segments.

Each single line segment can be edited separately from the other line segments in a series. 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 "Control the Properties of Objects" on page 263.

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" on page 312**
**"Use Object Snaps" on page 341**
**"Adjust Grid and Grid Snap" on page 347**
**"Draw Polylines" on page 391**
**"Offset an Object" on page 610**
**"Break and Join Objects" on page 634**

**To draw lines**

1. Click Draw menu ➤ Line.
2. Specify the start point.

   You can use the pointing device or enter coordinate values on the command line.
3. Complete the first line segment by specifying the endpoint.

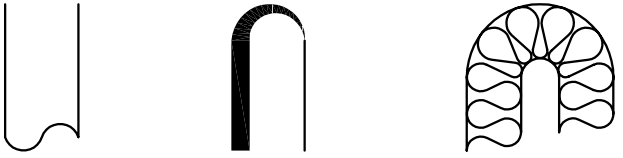
   To undo the previous line segment during the LINE command, enter **u** or click Undo on the toolbar.
4. Specify the endpoints of any additional line segments.
5. Press ENTER to end or **c** to close a series of line segments.

   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.

Draw toolbar

# Draw Polylines

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.



pipe symbol | differing widths | an insulated wall

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 or use to convert it to individual line and arc segments. You can

- Convert a spline-fit polyline into a true spline with
- 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

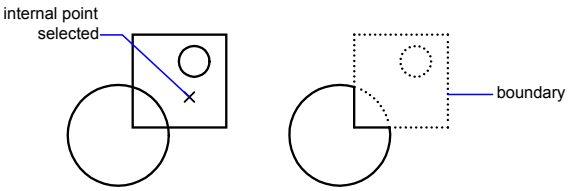
When you use the Width option, you are prompted for both a starting and an ending width. By entering different values, you can taper the polyline. The starting and ending points of wide polyline segments are in the center of the line. Intersections of adjacent wide segments are usually beveled. However, 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 define the boundary.




**See also:**


"Draw Rectangles and Polygons" on page 395
"Modify Complex Objects" on page 645
"Break and Join Objects" on page 634
"Control Lineweights" on page 301


### To draw a polyline with straight segments

1. Click Draw menu ➤ 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.

Draw toolbar

### To draw a line and arc combination polyline

1. Click Draw menu ➤ 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) on the command line.
- 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.

Draw toolbar

**To create a wide polyline**

1 Click Draw menu ➤ 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.

Draw toolbar

**To create a boundary polyline**

1 Click Draw menu ➤ 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.

# 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.

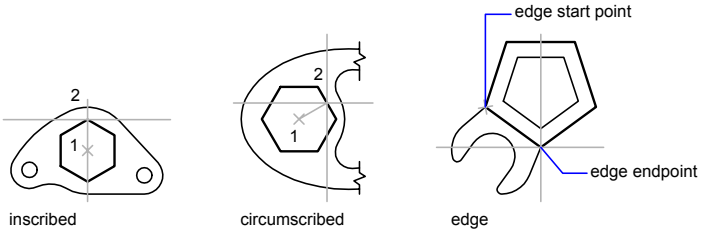
Two commands, RECTANG and POLYGON, provide an efficient method to create rectangles and regular polygons such as equilateral triangles, squares, pentagons, hexagons, and so on. If necessary, you can use to convert the resulting polyline object into lines.

### Draw Rectangles

Use to create closed polylines in a rectangular shape. You can specify the length, width, area, and rotation parameters. You can also control the type of corners on the rectangle—fillet, chamfer, or square.

### Draw Regular Polygons

Use 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.

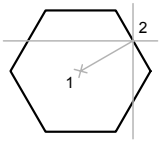



**See also:**

"Draw Polylines" on page 391

### To draw a circumscribed polygon

1. Click Draw menu ➤ Polygon.
2. On the command line, 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).




Draw toolbar

### To draw a polygon by specifying one edge

1. Click Draw menu ➤ Polygon.
2. On the command line, 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.

Draw toolbar

**To draw an inscribed polygon**

1 Click Draw menu ➤ Polygon.

2 On the command line, 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.

Draw toolbar

**To draw a rectangle**

1 Click Draw menu ➤ Rectangle.

2 Specify the first corner of the rectangle.

3 Specify the other corner of the rectangle.

Draw toolbar

# Draw Multiple-Line Objects

Multilines are composed of 1 to 16 parallel lines, called elements.

When you draw a multiline, you can use the STANDARD style, which has two elements, or specify a style that you created previously. You can also change the justification and scale of the multiline before you draw it.

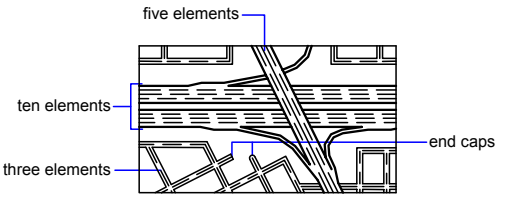
Multiline justification determines which side of the cursor that the multiline is drawn, or whether it is centered on the cursor.

Multiline scale controls the overall width of the multiline using the current units. Multiline scale does not affect linetype scale. If you change the multiline scale, you might need to make equivalent changes to the linetype scale to prevent dots or dashes from being disproportionately sized.

### Create Multiline Styles

You can create named styles for multilines to control the number of elements and the properties of each element. The properties of multilines include

- The total number of elements and position of each element
- The offset distance for each element from the middle of the multiline
- The color and linetype of each element
- The visibility of the lines, called *joints*, that appear at each vertex
- The type of end caps that are used
- The background fill color of the multiline




You can add up to 16 elements to a multiline style. Elements with a positive offset appear on one side of the middle of the multiline; elements with a negative offset appear on the other side of the middle of the multiline.

**See also:**

"Modify Multilines" on page 653

### To draw a multiline

1. Click Draw menu ➤ Multiline.
2. At the Command prompt, enter **st** to select a style.
3. To list available styles, enter the style name or enter **?**.
4. To justify the multiline, enter **j** and select top, zero, or bottom justification.

5 To change the scale of the multiline, enter **s** and enter a new scale.

Now draw the multiline.

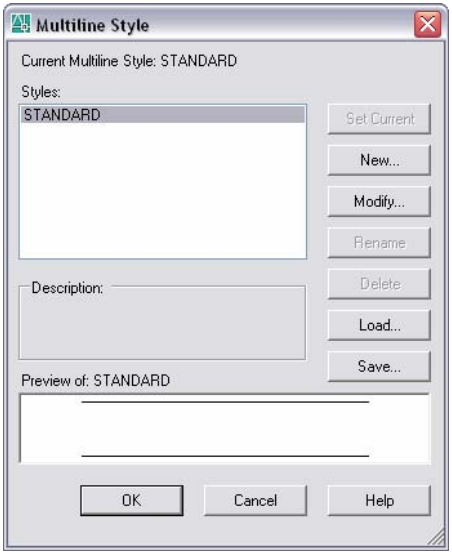
6 Specify the starting point.

7 Specify a second point.

8 Specify additional points, or press ENTER . If you specify three or more points, you can enter **c** to close the multiline.

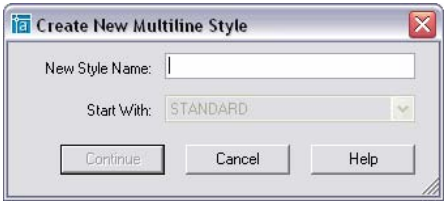
### To create a multiline style

1 Click Format menu ➤ Multiline Style.



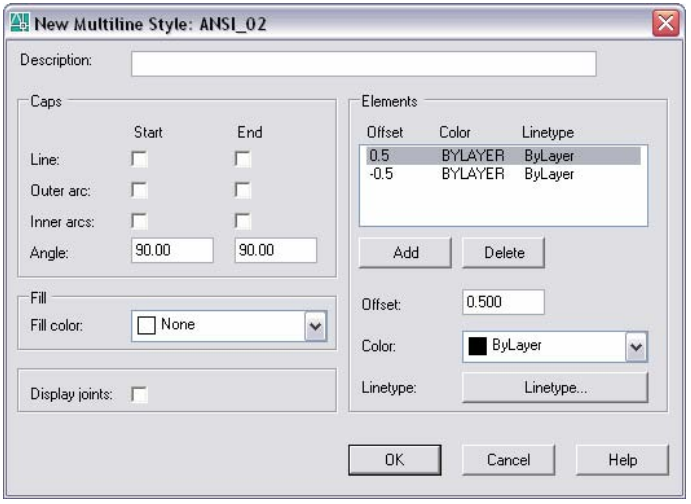

2 In the Multiline Style dialog box, click New.

3 In the Create New Multiline Style dialog box, enter a name for the multiline style and select a multiline style from which to start. Click Continue.



4 In the New Multiline Style dialog box, select the parameters for the multiline style. You can also enter a description.

Descriptions are optional and can be up to 255 characters, including spaces.




5 Click OK.

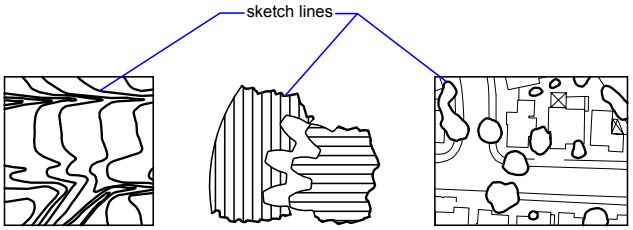
6 In the Multiline Style dialog box, click Save to save the multiline style to a file (the default is *acad.mln*). You can save multiline styles to the same file.

If you create more than one multiline style, save the current style before creating a new one or you lose the changes to the first style.

# Draw Freehand Sketches

Sketching is useful for creating irregular boundaries or for tracing with a digitizer.

You can use the  command to draw freehand sketches. Sketching is useful for creating irregular boundaries or for tracing with a digitizer.




### Create Sketches

To sketch, use the pointing device like a pen, clicking to put the "pen" down on the screen to draw and clicking again to lift it up and stop drawing. Freehand sketches comprise many line segments. Each line segment can be a separate object or a polyline. You set the minimum length or increment of the segments. Small line segments allow for greater accuracy, but they can greatly increase the drawing file size. For this reason, use this tool sparingly.

Before sketching, check the  system variable to make sure the current linetype is BYLAYER. If you use a linetype with dots or dashes and set the sketch line segment shorter than the spaces or dashes, you won't see the spaces or dashes.

### Erase Freehand Lines

You erase freehand lines by using the Erase option of the SKETCH command. In Erase mode, wherever the cursor intersects the freehand line, everything from the intersection to the end of the line is erased.

Once you record freehand lines, you can't edit them or erase them with the Erase option of SKETCH. Use the  command after you finish sketching.

### Sketch in Tablet Mode

You use Tablet mode with a digitizer. Sketching in Tablet mode is useful for such things as tracing map outlines from paper directly into a drawing. You can't turn off Tablet mode while sketching.

When Tablet mode is on, you can configure the program to map the paper drawing's coordinate system directly into the world coordinate system. Thus, there is a direct correlation between the coordinates where screen crosshairs appear, the coordinates on the tablet, and the coordinates in the original paper drawing. After configuring the program to match the coordinates of the paper drawing, you may find that the area shown on the screen is not the area you need. To avoid this problem, use  to display the entire work area before you start to sketch.

With some digitizers you can't select the menus while Tablet mode is on. See your digitizer documentation for details.

### Maintain Sketching Accuracy

To ensure accuracy on a slow computer, set the record increment value to a negative value. SKETCH uses this value as if it was positive but tests every point received from the pointer against twice the record increment. If the point is more than two record increments away, your computer beeps as a warning that you should slow down to avoid losing accuracy. For example, if the record increment is –1, you should move the cursor in increments of no more than 2. Using this method does not slow down the tracing speed.

### To sketch and record freehand lines

1. At the Command prompt, enter **sketch**.
2. At the Record Increment prompt, enter the minimum line segment length.
3. Click the start point to put the "pen" down.

   When you move the pointing device, temporary freehand line segments of the length you specified are drawn. SKETCH doesn't accept coordinate input. During the command, freehand lines are displayed in a different color.
4. Click the endpoint to lift the "pen" up so that you can move the cursor around the screen without drawing. Click a new start point to resume drawing from the new cursor position.
5. Enter **r** at any time to record (save) the line you're drawing in the database and those already drawn.

   If the pen is down, you can continue drawing after recording. If the pen is up, click to resume drawing. The freehand line starts from wherever the cursor is when you click.
6. Press ENTER to complete the sketch and record all unrecorded lines.

### To erase freehand lines

1. While running the command, with the pen up or down, enter **e** (Erase).

   If the pen was down, it moves up.
2. Move the cursor to the end of the line you drew last and then move it back as far along the line as you want to erase.
3. To end the erasure and return to the SKETCH Command prompt, enter **p**. To undo the erasure, enter **e**.

If you want to change the current viewport while sketching, make sure the pen is up, all lines entered so far have been recorded, and Tablet mode is off.

# Draw Curved Objects

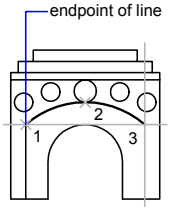
Curved objects are arcs, circles, polyline arcs, donuts, ellipses, and splines.

## Draw Arcs

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.

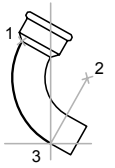
### Draw Arcs by Specifying Three Points

You can draw an arc by specifying three points. In the following example, the start point of the arc snaps to the endpoint of a line. The second point of the arc snaps to the middle circle in the illustration.

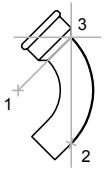



### Draw Arcs by Specifying Start, Center, End

When you know the start point, center point, and endpoint, you can draw an arc by specifying either the start point or the center point first. The center point is the center of a circle that the arc is part of.



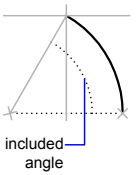

start(1), center(2), end(3)



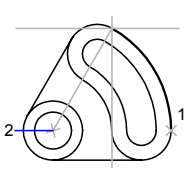

center(1), start(2), end(3)

### Draw Arcs by Specifying Start, Center, Angle

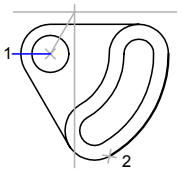
When you have a start point and a center point you can snap to and you know the included angle, use the Start, Center, Angle or the Center, Start, Angle option.



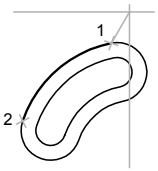

The included angle determines the endpoint of the arc. Use the Start, End, Angle method when you know both endpoints but cannot snap to a center point.




start, center, angle



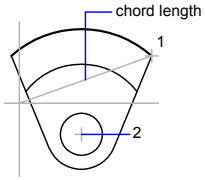

center, start, angle



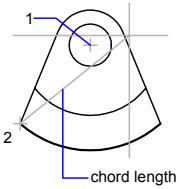

start, end, angle

### Draw Arcs by Specifying Start, Center, Length

When you have a start point and a center point you can snap to; when you know the chord length, use the Start, Center, Length or the Center, Start, Length option.




start, center, length



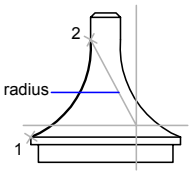

center, start, length

The length of the chord of the arc determines the included angle.

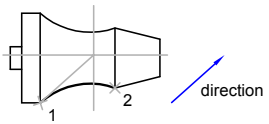
### Draw Arcs by Specifying Start, End, Direction/Radius

When you have a start point and an endpoint, use the Start, End, Direction or the Start, End, Radius option.

The illustration on the left shows an arc drawn by specifying a start point, endpoint, and radius. You can specify the radius by entering a length or by moving the pointing device clockwise or counterclockwise and clicking to specify a distance.




start, end, radius

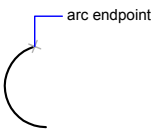


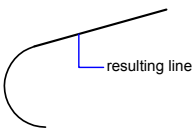

start, end, direction

The illustration on the right shows an arc drawn with the pointing device by specifying a start point and an endpoint and a direction. Moving the cursor up from the start point and endpoint draws the arc concave to the object, as shown here. Moving the cursor down draws the arc convex to the object.

### Draw Contiguous Arcs and Lines

Immediately after you complete an arc, you can start a line tangent to the arc at an endpoint by starting the  command and pressing  ENTER  at the Specify First Point prompt. You need to specify only the line length.




Conversely, after you complete a line, you can start an arc tangent to the line at an endpoint by starting the  command and pressing  ENTER  at the Specify Start Point prompt. You need to specify only the endpoint of the arc.

You can connect sequentially drawn arcs in the same way. To create connected arcs using a menu, click Arc from the Draw menu, and then click Continue. In both cases, the resulting object is tangent to the previous one. You can use the shortcut menu to repeat the Continue option.

**See also:**

"Draw Polylines" on page 391
"Break and Join Objects" on page 634

### To draw an arc by specifying three points

1. Click Draw menu ➤ Arc ➤ 3 Points.
2. Specify the start point.
3. Specify a point on the arc.
4. Specify the endpoint.

Draw toolbar

### To draw an arc using a start point, a center point, and an endpoint

1. Click Draw menu ➤ Arc ➤ Start, Center, End.
2. Specify a start point.
3. Specify the center point.
4. Specify the endpoint.

Draw toolbar

### To continue an arc with a tangential line

1. Complete the arc.
2. Click Draw menu ➤ Line.
3. Press ENTER at the first prompt.
4. Enter the length of the line and press ENTER .

Draw toolbar

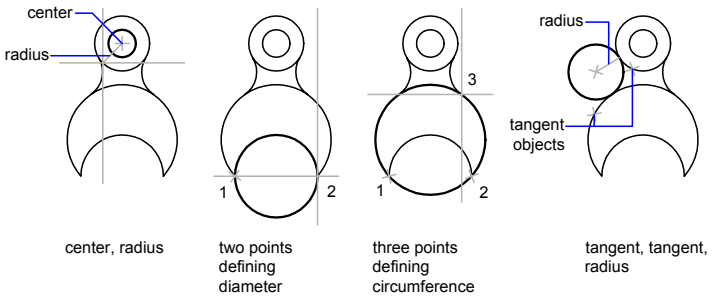
### To continue an arc with a tangential arc

1. Complete the arc.
2. Click Draw menu ➤ Arc ➤ Continue.
3. Specify the second endpoint of the tangent arc.

Draw toolbar

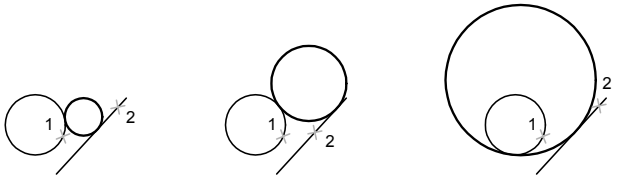
# Draw Circles

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 () to Tangent and use the three-point method to create the circle.

**See also:**

"Use Object Snaps" on page 341
"Draw Isometric Circles"

### To draw a circle by specifying a center point and radius or diameter

1. Click Draw menu ➤ Circle ➤ Center, Radius (or Center, Diameter).
2. Specify the center point.
3. Specify the radius or diameter.

Draw toolbar

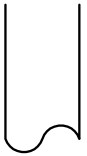
### To create a circle tangent to two objects

1. Click Draw menu ➤ Circle ➤ 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.

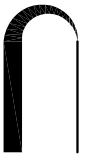
Draw toolbar

# Draw Polyline Arcs

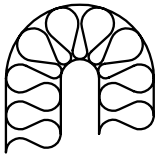
A polyline is a connected sequence of line segments created as a single object. You can create straight line segments, arc segments, or a combination of the two.



pipe symbol



differing widths



an insulated wall

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  or use  to convert it to individual line and arc segments. You can

- Convert a spline-fit polyline into a true spline with
- 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.

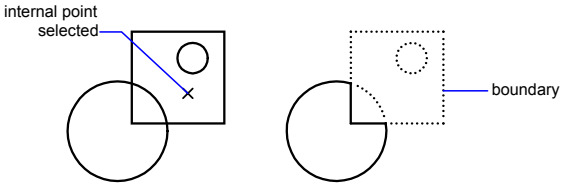
### Taper

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 ar not beveled.

### Create Polylines from the Boundaries of Objects

You can create a polyline from the boundaries of overlapping objects that form a closed area. A polyline created using the boundary method is a separate object, distinct from the objects used to create it. You can edit it using the same methods used to edit other polylines.

To expedite the boundary selection process in large or complex drawings, you can specify a group of boundary candidates, called a boundary set. You create this set by selecting the objects you want to use to define the boundary.




**See also:**

"Modify Splines" on page 650
"Modify or Join Polylines" on page 647
"Break and Join Objects" on page 634
"Control Lineweights" on page 301

### To draw a polyline with straight segments

1. Click Draw menu ➤ 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.

Draw toolbar

### To draw a line and arc combination polyline

1. Click Draw menu ➤ 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) on the command line.
   - 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.

Draw toolbar

### To create a wide polyline

1. Click Draw menu ➤ 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.

Draw toolbar

**To create a boundary polyline**

1 Click Draw menu ➤ Boundary.

2 In the Boundary Creation dialog box, in the Object Type list, select Polyline.

3 Under Boundary Set, do one of the following:

- To create a boundary set from all objects visible in the current viewport, select Current Viewport in the list. Avoid this option for large, complex drawings.
- To specify which objects to include in the boundary set, click New. Select the objects that you want to use to create the boundary. Choosing this option automatically selects the Existing Set option.

4 Click Pick Points.

5 Specify points within each area that you want to form a boundary polyline.

This area must be totally enclosed; that is, there can be no gaps between enclosing objects. You can select more than one area.

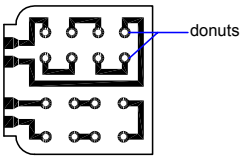
6 Press ENTER to create the boundary polyline and end the command.

The command creates a polyline in the shape of the boundary. Because this polyline overlaps the objects used to create it, it may not be visible. However, you can move, copy, or modify it just as you can any other polyline.

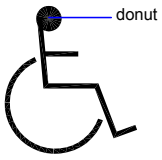
# Draw Donuts

Donuts are filled rings or solid-filled circles that actually are closed polylines with width.

To create a donut, you specify its inside and outside diameters and its center. You can continue creating multiple copies with the same diameter by specifying different center points. To create solid-filled circles, specify an inside diameter of 0.



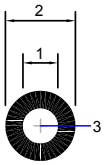

filled rings




solid-filled circles

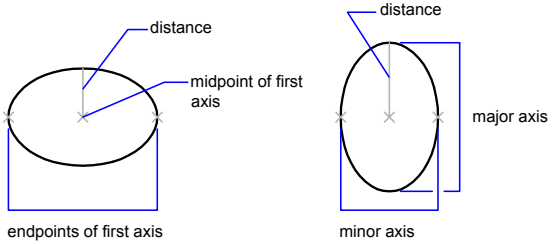
### To create a donut

1. Click Draw menu ➤ 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.

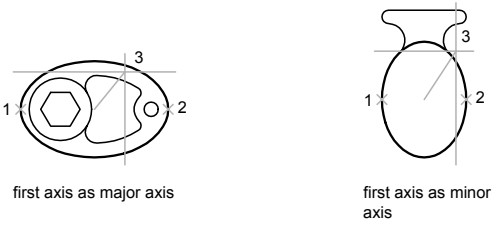


# Draw Ellipses

The shape of an ellipse is determined by two axes that define its length and width. The longer axis is called the major axis, and the shorter one is the minor axis.




The illustrations below show two different ellipses created by specifying axis and distance. The third point specifies only a distance and does not necessarily designate the axis endpoint.




If you are drawing on isometric planes to simulate 3D, you can use ellipses to represent isometric circles viewed from an oblique angle. First you need to turn on Isometric Snap in the Drafting Settings dialog box.

**See also:**

**"Draw Isometric Circles"**
**"Break and Join Objects" on page 634**

**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 Draw menu ➤ Ellipse ➤ Axis, End.

4 Enter **i** (Isocircle).

5 Specify the center of the circle.

6 Specify the radius or diameter of the circle.

Draw toolbar

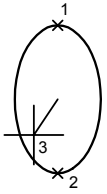
**To draw a true ellipse using endpoints and distance**

1 Click Draw menu ➤ Ellipse ➤ 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.




Draw toolbar

**To draw an elliptical arc using start and end angles**

1 Click Draw menu ➤ Ellipse ➤ Arc.

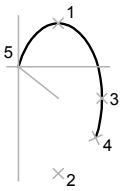
2 Specify endpoints for the first axis (1 and 2).

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.

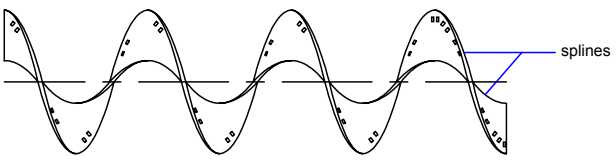



Draw toolbar

# Draw Splines

A spline is a smooth curve that passes through or near a given set of points. You can control how closely the curve fits the points.

The  command creates a particular type of spline known as a nonuniform rational B-spline (NURBS) curve. A NURBS curve produces a smooth curve between control points.




You create splines by specifying points. You can close the spline so that the start and endpoints are coincident and tangent.

Tolerance describes how closely the spline fits the set of fit points you specify. The lower the tolerance, the more closely the spline fits the points. At zero tolerance, the spline passes through the points. You can change the spline-fitting tolerance while drawing the spline to see the effect.

You can use two methods for creating splines:

- Create spline curves with the Spline option of  to smooth existing polylines created with . Such spline-fit polylines are created with uniform knot vectors and are more likely to be included in drawings created with earlier versions of the product.
- Create splines, which are NURBS curves, with . Drawings containing splines use less memory and disk space than those containing spline-fit polylines of similar shape.

You can easily convert spline-fit polylines into true splines with SPLINE.

**See also:**

"Modify Splines" on page 650
"Break and Join Objects" on page 634

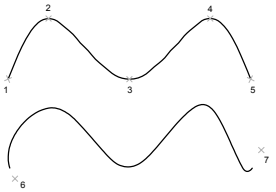
### To convert a spline-fit polyline to a spline

1. Click Draw menu ➤ Spline.
2. Enter **o** (Object).
3. Select a spline-fit polyline and press  ENTER .

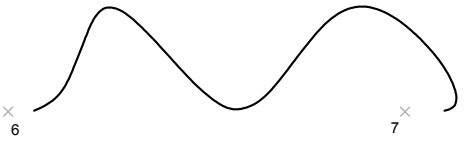
   The selected object changes from a polyline to a spline.

### To convert a spline by specifying points

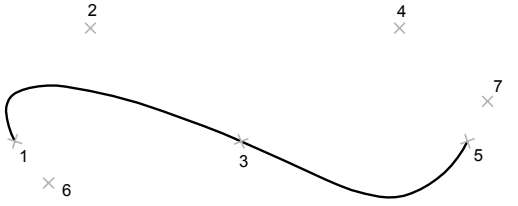
1. Click Draw menu ➤ Spline.
2. Specify the start point for the spline (1).
3. Specify points (2 through 5) to create the spline, and press  ENTER .
4. Specify the start and end tangents (6, 7).




The spline below uses the same points but different start and end tangents.




The spline below uses the same points but a higher tolerance and different start and end tangents.




Draw toolbar

# 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  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 Draw menu ➤ Point ➤ Single Point.
2. Specify the point location.

   You can snap to a point using the Node object snap.

Draw toolbar

# 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.

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 (xline) can be placed anywhere in three-dimensional space. You can specify its orientation in several ways. The default method for creating the line is the two-point method: you specify two points to define the

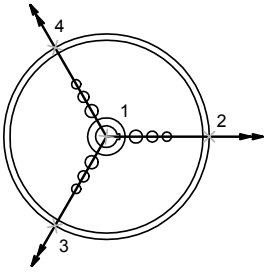
orientation. The first point, the root, is the conceptual midpoint of the construction line, that is, the point snapped to by the Midpoint object snap.

You can also create construction lines in several other ways.

- **Horizontal and Vertical**. Create construction lines that pass through a point you specify and are parallel to the *X* or *Y* axis of the current UCS.
- **Angle**. Creates a construction line in one of two ways. Either you select a reference line and then specify the angle of the construction line from that line, or you create a construction line at a specific angle to the horizontal axis by specifying an angle and then a point through which the construction line should pass.
- **Bisector**. Creates a construction line that bisects an angle you specify. You specify the vertex and the lines that create the angle.
- **Offset**. Creates a construction line parallel to a baseline you specify. You specify the offset distance, select the baseline, and then indicate on which side of the baseline to locate the construction line.

#### Rays

A ray is a line in three-dimensional space that starts at a point you specify and extends to infinity. Unlike construction lines, which extend in two directions, rays extend in only one direction. Using rays instead of construction lines can help reduce visual clutter. Like construction lines, rays are ignored by commands that display the drawing extents.



three rays

#### To create a construction line by specifying two points

1. Click Draw menu ➤ 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.

Draw toolbar

**To create a ray**

1 Click Draw menu ➤ 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.

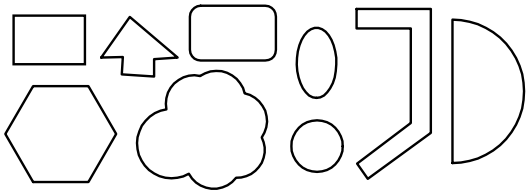
# Create and Combine Areas (Regions)

Regions are two-dimensional enclosed areas that have physical properties such as centroids or centers of mass. You can combine existing regions into a single, complex region to calculate area.

Regions are two-dimensional enclosed areas you create from objects that form closed loops. Loops can be combinations of lines, polylines, circles, arcs, ellipses, elliptical arcs, and splines. The objects that make up the loops must either be closed or form closed areas by sharing endpoints with other objects.

Regions can be used for

- Applying hatching and shading
- Analyzing properties, such as area, using
- Extracting design information, such as the centroid



shapes that can form regions

You can create regions out of multiple loops and out of open curves whose endpoints are connected and form loops. You cannot form regions from open objects that intersect to form a closed area: for example, intersecting arcs or self-intersecting curves.

You can also create regions using .

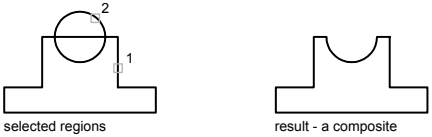
You create composite regions by combining, subtracting, or finding the intersection of regions. After forming these more complex regions, you can apply hatching or analyze their area.

Objects combined using :

selected regions

result

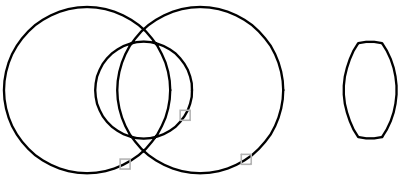
Objects combined using :




selected regions

result - a composite region

Objects combined using :




### To define regions

1. Click Draw menu ➤ 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 on the command line indicates how many loops were detected and how many regions were created.

### To define regions by using boundaries

1. Click Draw menu ➤ Boundary.
2. In the Boundary Creation dialog box, in the Object Type list, select Region.
3. Click Pick Points.
4. Specify a point in your drawing inside each closed area that you want to define as a region and press ENTER .

   This point is known as the internal point.

   **NOTE** You can make a new boundary set to limit the objects used to determine the boundary.

### To combine regions by adding

1. Click Modify menu ➤ Solids Editing ➤ 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 Modify menu ➤ Solids Editing ➤ Subtract.

2 Select one or more regions from which to subtract and press ENTER .

3 Select the region to subtract and press ENTER .

The areas of the second regions you selected are subtracted from the areas of the first regions.

**To combine regions by finding intersections**

1 Click Modify menu ➤ Solids Editing ➤ 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.

# Create 3D Objects

3D objects can be represented by simulated surfaces (3D thickness), as a wireframe model, as a surface model, or as a solid model.

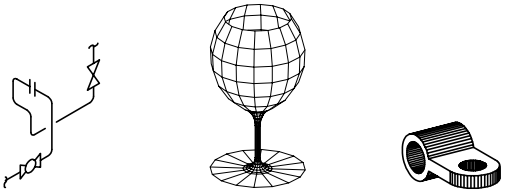
## Overview of 3D Objects

Although 3D models can be more difficult and time-consuming to create than 3D views of 2D objects, 3D modeling has several advantages. You can

- View the model from any vantage point
- Generate reliable standard and auxiliary 2D views automatically
- Create 2D profiles ()
- Remove hidden lines and do realistic shading
- Check interference

- Export the model to create an animation
- Perform engineering analysis
- Extract manufacturing data

Three types of 3D modeling are supported: wireframe, surface, and solid. Each type has its own creation and editing techniques.




A wireframe model is a skeletal description of a 3D object. There are no surfaces in a wireframe model; it consists only of points, lines, and curves that describe the edges of the object. You can create wireframe models by positioning 2D (planar) objects anywhere in 3D space. Some 3D wireframe objects are also provided, such as 3D polylines (that can only have a CONTINUOUS linetype) and splines. 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.

Surface modeling is more sophisticated than wireframe modeling in that it defines not only the edges of a 3D object, but also its surfaces. The surface modeler defines faceted surfaces using a polygonal mesh. Because the faces of the mesh are planar, the mesh can only approximate curved surfaces. With Autodesk Mechanical Desktop®, you can create true curved surfaces. To differentiate these two types of surfaces, faceted surfaces are called meshes.

Solid modeling is the easiest type of 3D modeling to use. With the solid modeler, you can make 3D objects by creating basic 3D shapes: boxes, cones, cylinders, spheres, wedges, and tori (donuts). You can then combine these shapes to create more complex solids by joining or subtracting them or finding their intersecting (overlapping) volume. You can also create solids by sweeping a 2D object along a path or revolving it about an axis. With Autodesk Mechanical Desktop, you can also define solids parametrically and maintain associativity between 3D models and the 2D views that you generate from them.

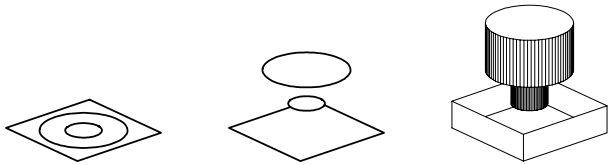
**NOTE** Because each modeling type uses a different method for constructing 3D models and editing methods vary in their effect on the different model types, it is recommended that you not mix modeling methods. Limited conversion between model types is available from solids to surfaces and from surfaces to wireframes; however, you cannot convert from wireframes to surfaces or from surfaces to solids.

# Add 3D Thickness to Objects

Thickness is a property of certain objects that gives them a 3D appearance.

Thickness is a property of certain objects that gives them 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 *Z* direction is determined by the orientation of the UCS at the time the object was created. 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  system variable. Change the thickness property of an existing object using the Properties palette. The 3D thickening is applied uniformly on an object: a single object cannot have different thicknesses for its various points.

You may 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. On the command line, enter the value for the thickness distance.

   New objects are created with 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 now display the specified 3D thickness.

Standard toolbar

# Create Wireframe Models

A wireframe model is an edge or skeletal representation of a real-world 3D object using lines and curves.

A wireframe model is an edge or skeletal representation of a real-world 3D object using lines and curves. You can use a wireframe model to

■ View the model from any vantage point

■ Generate standard orthographic and auxiliary views automatically

■ Generate exploded and perspective views easily

- Analyze spatial relationships, including the shortest distance between corners and edges, and checking for interferences
- Reduce the number of prototypes required

### 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 construction plane to orient planar objects such as circles and arcs. The UCS also determines the plane of operation for trimming and extending, offsetting, and rotating objects.
- Use object snaps and grid snap carefully to ensure the precision of your model.
- Use coordinate filters to drop perpendiculars and easily locate points in 3D based on the location of points on other objects.

### Methods for Creating Wireframe Models

You can create wireframe models by positioning any 2D planar object anywhere in 3D space, using the following methods:

- Entering 3D coordinates. You enter coordinates that define the *X, Y,* and *Z* location of the object.
- Setting the default construction plane (the *XY* plane of the UCS) on which you will draw the object.
- Moving or copying the object to its proper 3D location after you create it.

Wireframe modeling is a skill that requires practice and experience. The best approach to learning how to create wireframe models is to begin with simple models before attempting models that are more complex.

**To drop a perpendicular line from a 3D point down to the XY plane**

1. Click Draw menu ➤ Line.
2. Use an object snap to specify a point on an object that is not on the *XY* plane of the UCS.

   This 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.

   Using coordinate filters, 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.

# Create Surfaces

The surface modeler creates faceted surfaces using a polygonal mesh. Because the faces of the mesh are planar, the mesh can only approximate curved surfaces.

Use surface meshes if you need hiding, shading, and rendering capabilities that wireframe models cannot provide but do not need the physical properties that solid models provide (mass, volume, center of gravity, moments of inertia, and so on). Meshes are useful if you want to create geometry with unusual mesh patterns, such as a 3D topographical model of mountainous terrain.

Surface meshes are displayed as wireframe representations until you use , , or . Use REGEN (after using HIDE) and SHADEMODE to restore the wireframe display.

You can create several types of surfaces:

- *3D face*. creates a planar surface with either three or four sides.
- *Ruled surface*. creates a polygon mesh representing the ruled surface between two lines or curves.
- *Tabulated surface*. creates a polygon mesh representing a general tabulated surface defined by the extrusion of a line or curve (called a path curve) in a specified direction and distance (called a direction vector).
- *Revolved surface*. creates a polygon mesh approximating a surface of revolution by rotating a path curve or profile (lines, circles, arcs, ellipses,

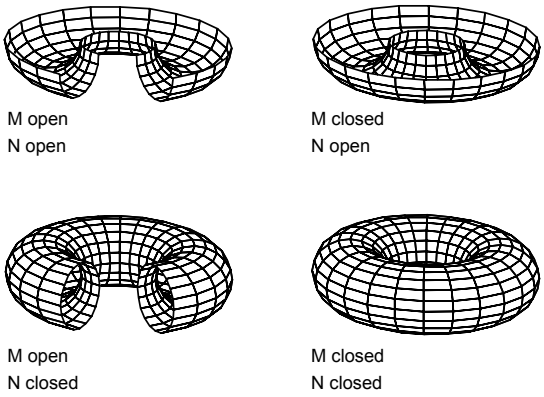
elliptical arcs, polylines, or splines, closed polylines, polygons, closed splines, or donuts) about a specified axis.

- *Edge-defined surface*. creates a polygon mesh approximating a Coons surface patch mesh from four adjoining edges. A Coons surface patch mesh is a bicubic surface interpolated between four adjoining edges (which can be general space curves).
- *Predefined 3D surface*. creates three-dimensional polygon mesh objects in common geometric shapes, including boxes, cones, spheres, tori, wedges, and pyramids.
- *General surface meshes*. and PFACE create three-dimensional polygon mesh objects in any shape.

### Understand Mesh Construction

The mesh density controls the number of facets on a surface, and is defined in terms of a matrix of *M* and *N* vertices, similar to a grid consisting of columns and rows. *M* and *N* specify the column and row position, respectively, of any given vertex.

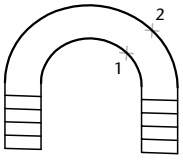
A mesh can be open or closed. A mesh is open in a given direction if the start and end edges of the mesh do not touch, as shown in the following illustrations.



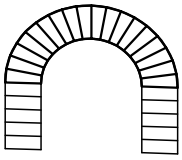

There are several methods for creating meshes.

### Create a Ruled Surface Mesh

With , you can create a surface mesh between two lines or curves. You can use two different objects to define the edges of the ruled surface: lines, points, arcs, circles, ellipses, elliptical arcs, 2D polylines, 3D polylines, or splines. Pairs of objects to be used as the "rails" of a ruled surface mesh must both be either open or closed. You can pair a point object with either an open or a closed object.

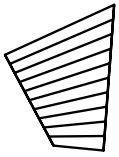


curves defined

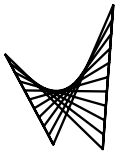

result

You can specify any two points on closed curves to complete RULESURF. For open curves, construction of the ruled surface is based on the locations of the specified points on the curves.

specified point on
corresponding sides


result

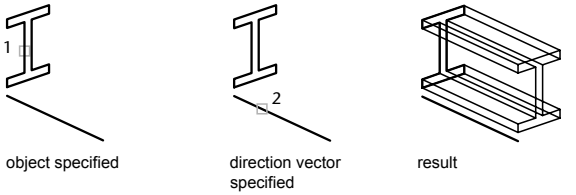
specified points on
opposite sides


result

### Create a Tabulated Surface Mesh

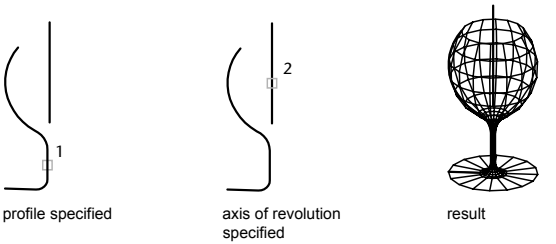
With the  command, you can create a surface mesh representing a general tabulated surface defined by a path curve and a direction vector. The path

curve can be a line, arc, circle, ellipse, elliptical arc, 2D polyline, 3D polyline, or spline. The direction vector can be a line or an open 2D or 3D polyline. TABSURF creates the mesh as a series of parallel polygons running along a specified path. You must have the original object and the direction vector already drawn, as shown in the following illustrations.




### Create a Revolved Surface Mesh

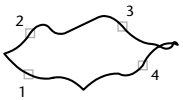
Use the  command to create a revolved surface, also called a surface of revolution, by rotating a profile of the object about an axis. REVSURF is useful for surfaces with rotational symmetry.



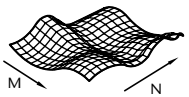

The profile is called a path curve, which can be any combination of lines, circles, arcs, ellipses, elliptical arcs, polylines, or splines, closed polylines, polygons, closed splines, or donuts.

### Create an Edge-Defined Surface Mesh

With the  command, you can create a *Coons surface patch* mesh, as shown in the following illustration, from four objects called *edges*. Edges can be arcs, lines, polylines, splines, and elliptical arcs, and they must form a closed loop and share endpoints. A Coons patch is a bicubic surface (one curve in the *M* direction and another in the *N* direction) interpolated between the four edges.



four edges selected



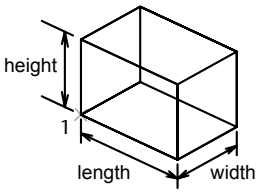
result

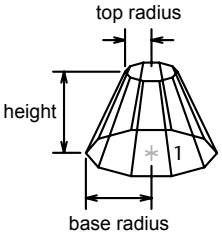
### Create a Predefined 3D Surface Mesh

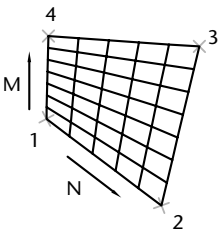
The 3D command creates the following 3D shapes: boxes, cones, dishes, domes, meshes, pyramids, spheres, tori (donuts), and wedges.

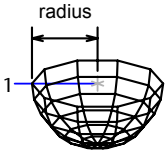
To view the objects you are creating with the 3D command more clearly, set a viewing direction with , , or .

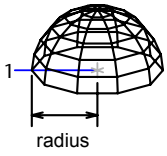
In the following illustrations, the numbers indicate points you specify to create the mesh.

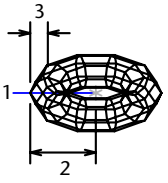


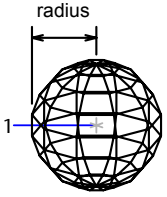



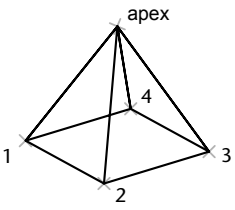



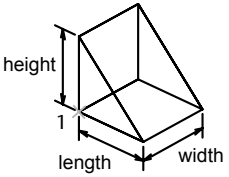



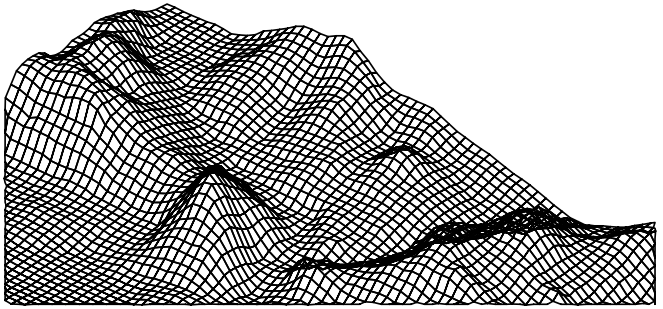



### Create a Rectangular Mesh

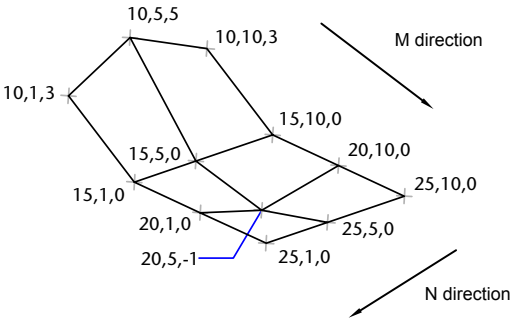
With the  command, you can create polygon meshes that are open in both the *M* and *N* directions (similar to the *X* and *Y* axes of an *XY* plane). You can close the meshes with . You can use 3DMESH to construct very irregular surfaces. In most cases, you can use 3DMESH in conjunction with scripts or AutoLISP routines when you know the mesh points.



In the following example of text on the command line, you enter the coordinate values for each vertex to create the mesh in the illustration.

**Example:**

Command: **3dmesh**
Mesh M size: **4**
Mesh N size: **3**
Vertex (0, 0): **10,1, 3**
Vertex (0, 1): **10, 5, 5**
Vertex (0, 2): **10,10, 3**
Vertex (1, 0): **15,1, 0**
Vertex (1, 1): **15, 5, 0**
Vertex (1, 2): **15,10, 0**
Vertex (2, 0): **20,1, 0**
Vertex (2, 1): **20, 5, –1**
Vertex (2, 2): **20,10 ,0**
Vertex (3, 0): **25,1, 0**
Vertex (3, 1): **25, 5, 0**
Vertex (3, 2): **25,10, 0**




### Create a Polyface Mesh

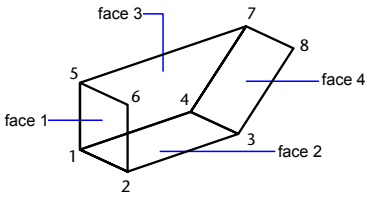
The  command produces a polyface (polygon) mesh, with each face capable of having numerous vertices. PFACE is typically used by applications rather than by direct user input.

Creating a polyface mesh is similar to creating a rectangular mesh. To create a polyface mesh, you specify coordinates for its vertices. You then define each face by entering vertex numbers for all the vertices of that face. As you create the polyface mesh, you can set specific edges to be invisible, assign them to layers, or give them colors.

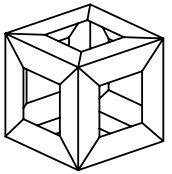
To make the edge invisible, enter the vertex number as a negative value. For instance, to make the edge between vertices 5 and 7 invisible in the following illustration, you enter the following:

Face 3, vertex 3: **–7**

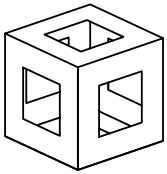
In the illustration, face 1 is defined by vertices 1, 5, 6, and 2. Face 2 is defined by vertices 1, 4, 3, and 2. Face 3 is defined by vertices 1, 4, 7, and 5, and face 4 is defined by vertices 3, 4, 7, and 8.



You can control the display of invisible edges with the  system variable. If SPLFRAME is set to a nonzero value, the invisible edges become visible and can then be edited. If SPLFRAME is set to 0, the invisible edges remain invisible.



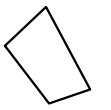
SPLFRAME = 1



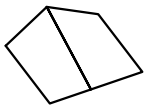
SPLFRAME = 0

**To create a rectangular mesh**

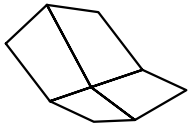
1. Click Draw menu ➤ Surfaces ➤ 3D Mesh.
2. Specify the *M* size, using an integer from 2 through 256.
3. Specify the *N* size, using an integer from 2 through 256.
4. Specify the vertex points as prompted. Specifying the last vertex point completes the mesh.



Mesh M size: 2
Mesh N size: 2



Mesh M size: 2
Mesh N size: 3



Mesh M size: 3
Mesh N size: 3

Surfaces toolbar

**To create a ruled surface**

1. Click Draw menu ➤ Surfaces ➤ Ruled Surface.
2. Select the first defining curve. Then select the second.
3. Erase the original curve if necessary.

Surfaces toolbar

**To create a tabulated surface mesh**

1. Click Draw menu ➤ Surfaces ➤ Tabulated Surface.
2. Specify a path curve.
3. Specify a direction vector.
4. Erase the original objects if necessary.

Surfaces toolbar

**To create a surface of revolution mesh**

1. Click Draw menu ➤ Surfaces ➤ Revolved Surface.
2. Specify a path curve.

   The path curve, which defines the *N* direction of the mesh, can be a line, arc, circle, ellipse, elliptical arc, 2D polyline, 3D polyline, or spline. If you select a circle, closed ellipse, or closed polyline, the mesh is closed in the *N* direction.
3. Specify the axis of revolution.

   The direction vector can be a line or an open 2D or 3D polyline. If you choose a polyline, the vector sets the rotation axis from its first vertex to its last vertex. Any intermediate vertices are ignored. The axis of revolution determines the *M* direction of the mesh.
4. Specify the start angle. Then specify the included angle.

   If you specify a nonzero start angle, the mesh is generated at a position offset from the path curve by that angle. The included angle specifies how far about the axis of revolution the surface should extend.
5. Erase the original objects if necessary.

Surfaces toolbar

**To create an edge-defined Coons surface patch mesh**

1. Click Draw menu ➤ Surfaces ➤ Edge Surface.
2. Select the four edges in any order.

   The first edge you select determines the mesh's *M* direction.

Surfaces toolbar

# Create 3D Solids

A solid object represents the entire volume of an object. Complex solid shapes are easier to construct and edit than wireframes and meshes.

A solid object represents the entire volume of an object. Solids are the most informationally complete and least ambiguous of the 3D modeling types. Complex solid shapes are also easier to construct and edit than wireframes and meshes.

You create solids from one of the basic solid shapes of box, cone, cylinder, sphere, torus, and wedge or by extruding a 2D object along a path or revolving a 2D object about an axis.

Once you have created a solid in this manner, you can create more complex shapes by combining solids. You can join solids, subtract solids from each other, or find the common volume (overlapping portion) of solids.

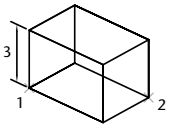
Solids can be further modified by filleting, chamfering, or changing the color of their edges. Faces on solids are easily manipulated because they don't require you to draw any new geometry or perform Boolean operations on the solid. This program also has commands for slicing a solid into two pieces or obtaining the 2D cross section of a solid (see "Modify 3D Solids" on page 655).

Like meshes, solids are displayed as wireframes until you hide, shade, or render them. Additionally, you can analyze solids for their mass properties (volume, moments of inertia, center of gravity, and so on). You can export data about a solid object to applications such as NC (numerical control) milling or FEM (finite element method) analysis. By exploding a solid, you can break it down to mesh and wireframe objects.

The  system variable controls the number of tessellation lines used to visualize curved portions of the wireframe. The  system variable adjusts the smoothness of shaded and hidden-line objects.

### Create a Solid Box

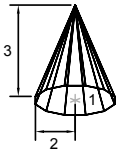
You can use  to create a solid box. The base of the box is always parallel to the *XY* plane of the current UCS.




The  or  command creates a rectangle or closed polyline from which you can create a box using . The  command creates a box shape defined by surfaces only.

### Create a Solid Cone

You can use  to create a solid cone defined by a circular or an elliptical base tapering to a point perpendicular to its base. By default, the cone's base lies on the *XY* plane of the current UCS. The height, which can be positive or negative, is parallel to the *Z* axis. The apex determines the height and orientation of the cone.

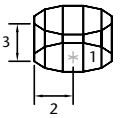


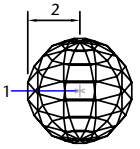

To create a truncated cone or a cone that requires a specific angle to define its sides, draw a 2D circle and then use EXTRUDE to taper the circle at an angle along the *Z* axis. To complete the truncation, you can subtract a box from the tip of the cone with the  command.  creates a circle from which you can create a cone using EXTRUDE with its Taper option. The 3D command creates a conical shape defined by surfaces only.

### Create a Solid Cylinder

You can use  to create a solid cylinder with a circular or an elliptical base. The base of the cylinder lies on the *XY* plane of the current UCS.

If you want to construct a cylinder with special detail, such as grooves along its sides, create a 2D profile of its base with a closed PLINE and use EXTRUDE to define its height along the *Z* axis. CIRCLE creates a circle from which you can create a cylinder using EXTRUDE.




To create a dome or dish, combine a sphere with a box and use SUBTRACT. If you want to create a spherical object that has additional detail, create a 2D profile and use  to define a rotation angle about the *Z* axis. The 3D command creates a spherical shape defined by surfaces only.

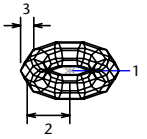
### Create a Solid Torus

You can use  to create a ring-shaped solid similar to the inner tube of a tire. The torus is parallel to and bisected by the *XY* plane of the current UCS. A torus is defined by two radius values, one for the tube and the other for the distance from the center of the torus to the center of the tube.

To create a lemon-shaped solid, use a negative torus radius and a positive number of greater magnitude for the tube radius. For example, if the torus radius is –2.0, the tube radius must be greater than 2.0.

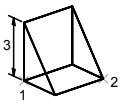
A torus may be self-intersecting. A self-intersecting torus has no center hole because the radius of the tube is greater than the radius of the torus.

The 3D command creates a toroidal shape defined by surfaces only.



### Create a Solid Wedge

You can use  to create a solid wedge. The base of the wedge is parallel to the *XY* plane of the current UCS with the sloped face opposite the first corner. Its height, which can be positive or negative, is parallel to the *Z* axis.



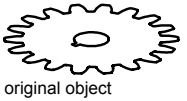

The 3D command creates a wedge shape defined by surfaces only.

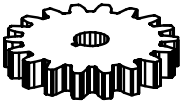
### Create an Extruded Solid

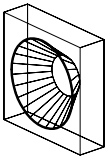
With EXTRUDE, you can create solids by extruding selected objects. You can extrude closed objects such as polylines, polygons, rectangles, circles, ellipses, closed splines, donuts, and regions. You cannot extrude 3D objects, objects contained within a block, polylines that have crossing or intersecting segments, or polylines that are not closed. You can extrude an object along a path, or you can specify a height value and a tapered angle.

Use EXTRUDE to create a solid from a common profile of an object, such as a gear or sprocket. EXTRUDE is particularly useful for objects that contain fillets, chamfers, and other details that might otherwise be difficult to reproduce except in a profile. If you create a profile using lines or arcs, use the Join option of  to convert them to a single polyline object or make them into a region before you use EXTRUDE.

Tapering the extrusion is useful specifically for parts that need their sides defined along an angle, such as a mold used to create metal products in a foundry. Avoid using extremely large tapered angles. If the angle is too large, the profile can taper to a point before it reaches the specified height.


original object

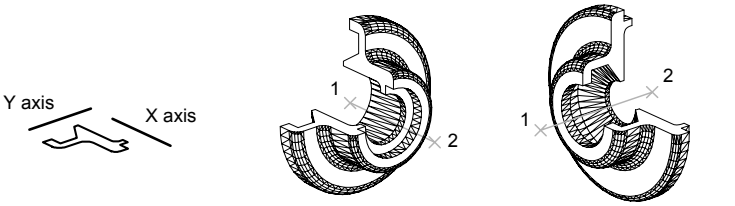

extruded object


extruded and
tapered circle

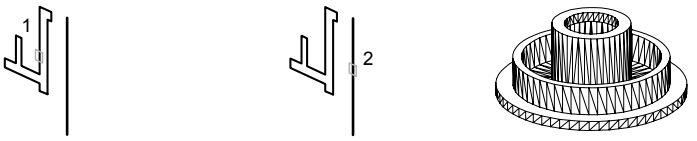
### Create a Revolved Solid

With REVOLVE, you can create a solid by revolving a closed object about the *X* or *Y* axis of the current UCS, using a specified angle. You can also revolve the object about a line, polyline, or two specified points. Similar to EXTRUDE, REVOLVE is useful for objects that contain fillets or other details that would otherwise be difficult to reproduce in a common profile. If you create a profile using lines or arcs that meet a polyline, use the PEDIT Join option to convert them to a single polyline object before you use REVOLVE.

You can use REVOLVE on closed objects such as polylines, polygons, rectangles, circles, ellipses, and regions. You cannot use REVOLVE on 3D objects, objects contained within a block, polylines that have crossing or intersecting segments, or polylines that are not closed.




original polyline | revolved about X axis | revolved about Y axis

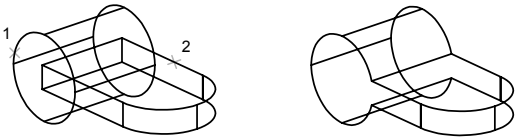



object to revolve selected | axis selected | result

### Create a Composite Solid

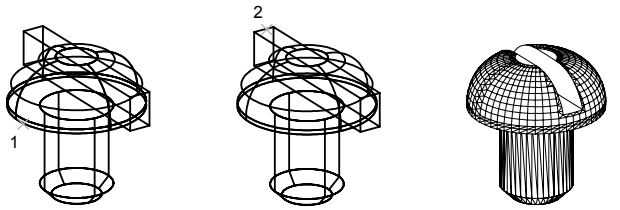
You can combine, subtract, and find the intersection of existing solids to create composite solids.

With , you can combine the total volume of two or more solids or two or more regions into a composite object.



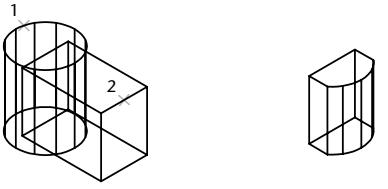

objects to be combined result

With SUBTRACT, you can remove the common area of one set of solids from another. For example, you can use SUBTRACT to add holes to a mechanical part by subtracting cylinders from the object.




objects to subtract from objects to subtract selected result (lines hidden for clarity)

With , you can create a composite solid from the common volume of two or more overlapping solids. INTERSECT removes the nonoverlapping portions and creates a composite solid from the common volume.




objects to intersect selected result

performs the same operation as INTERSECT, but INTERFERE keeps the original two objects.

**To create a solid box**

1 Click Draw menu ➤ Solids ➤ Box.

2 Specify the first corner of the base.

3 Specify the opposite corner of the base.

4 Specify the height.

**To create a solid cone with a circular base**

1 Click Draw menu ➤ Solids ➤ Cone.

2 Specify the base center point.

3 Specify the radius or diameter of the base.

4 Specify the height.

**To create a solid cone with an elliptical base**

1 Click Draw menu ➤ Solids ➤ Cone.

2 Enter **e** (Elliptical).

3 Specify an axis endpoint.

4 Specify a second axis endpoint.

5 Specify the length of the other axis.

6 Specify height, and then press ENTER .

**To create a solid cylinder with a circular base**

1 Click Draw menu ➤ Solids ➤ Cylinder.

2 Specify the base center point.

3 Specify the radius or diameter of the base.

4 Specify the height.

**To create a solid sphere**

1 Click Draw menu ➤ Solids ➤ Sphere.

2 Specify the center of the sphere.

3 Specify the radius or diameter of the sphere.

**To create a solid torus**

1 Click Draw menu ➤ Solids ➤ Torus.

2 Specify the center of the torus.

3 Specify the radius or diameter of the torus.

4 Specify the radius or diameter of the tube.

**To create a solid wedge**

1 Click Draw menu ➤ Solids ➤ Wedge.

2 Specify the first corner of the base.

3 Specify the opposite corner of the base.

4 Specify the height of the wedge.

**To extrude an object along a path**

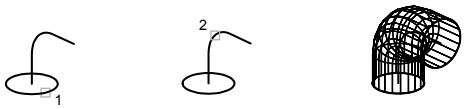
1 Click Draw menu ➤ Solids ➤ Extrude.

2 Select the objects to extrude.

3 Enter **p** (Path).

4 Select the object to use as the path.

After the extrusion, the original object may be deleted or retained, depending on the setting of the  system variable.




### To revolve an object about an axis

1 Click Draw menu ➤ Solids ➤ Revolve.

2 Select the objects to revolve.

3 Specify the start point and endpoint of the axis of revolution.

Specify the points so that the object is on one side of the axis points you specify. The positive axis direction is from the start point to the endpoint.

4 Specify the angle of revolution.

### To combine solids

1 Click Modify menu ➤ Solids Editing ➤ Union.

2 Select the objects to combine.

### To subtract one set of solids from another

1 Click Modify menu ➤ Solids Editing ➤ Subtract.

2 Select the objects to subtract from.

3 Select the objects to subtract.

**To create a solid from the intersection of two or more other solids**

1 Click Modify menu ➤ Solids Editing ➤ Intersect.

2 Select the objects to intersect.

# Create Revision Clouds

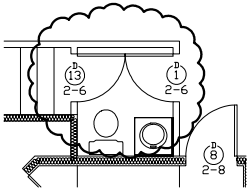
Revision clouds are polylines that consist of sequential arcs. They are used to call attention to parts of a drawing during the review stage.

If you review or redline drawings, you can increase your productivity by using the Revision Cloud feature to highlight your markups.  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  is set to 1 (the default).

You can set the minimum and maximum default values for the arc lengths of a revision cloud. When you draw a revision cloud, you can vary the size of the arcs by using pick points for the smaller arc segments. You can also edit the individual arc lengths and chord lengths of a revision cloud by adjusting the pick points.

REVCLOUD stores the last used arc length as a multiple of the  system variable to provide consistency among drawings with different scale factors.




Make sure that you can see the entire area to be outlined with REVCLOUD before you begin the command. REVCLOUD is not designed to support transparent and real-time panning and zooming.

**To create a revision cloud from scratch**

1. Click Draw menu ➤ 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.

Draw toolbar

**To create revision clouds with a calligraphy pen style**

1. Click Draw menu ➤ 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.

Draw toolbar

**To convert an object to a revision cloud**

1. Click Draw menu ➤ 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** on the command line and press ENTER.

4 Press ENTER to change the selected object to a revision cloud.

Draw toolbar

**To change the default values for arc lengths in a revision cloud**

1 Click Draw menu ➤ 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.

Draw toolbar

**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.

# Create and Use Blocks (Symbols)

# 20

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.

## In this chapter


- Overview of Blocks
- Create and Store Blocks
- Add Dynamic Behavior to Blocks
- Control the Color and Linetype Properties in Blocks
- Nest Blocks
- Insert Blocks
- Work with Dynamic Blocks in Drawings
- Attach Data to Blocks (Block Attributes)
- Modify Blocks
- Change the Color and Linetype in a Block
- Disassemble a Block Reference (Explode)
- Remove Block Definitions

# Overview of Blocks

You can use several methods to create blocks:

- Combine objects to create a block definition in your current drawing.
- Use the Block Editor to add dynamic behavior to a block definition in your current drawing.
- Create a drawing file and later insert it as a block in other drawings.
- Create a drawing file with several related block definitions to serve as a block library.

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.

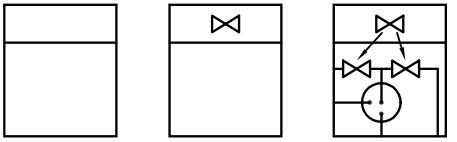
A block definition can also contain elements that add dynamic behavior to the block. You add these elements to the block in the Block Editor. When you add dynamic behavior to a block, you add flexibility and intelligence to the geometry. When you insert a block reference with dynamic behavior in a drawing, you can manipulate the geometry of the block reference through custom grips or custom properties, depending on how the block was defined.

You can use  to remove unused block definitions from a drawing.

# Create and Store Blocks

## How Blocks Are Stored and Referenced

Every drawing file has an invisible data area called the block definition table. The block definition table 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. The following illustrations are conceptual representations of three drawing files. Each rectangle represents a separate drawing file and is divided into two parts: the smaller part represents the block definition table, and the larger part represents the objects in a drawing.




When you insert a block, you are inserting a block reference. The information is not simply copied from the block definition to the drawing area. Instead, a link is established between the block reference and the block definition. Therefore, if the block definition is changed, all references are updated automatically.

To reduce the size of a drawing, you can purge unused block definitions.

No procedures for this topic.

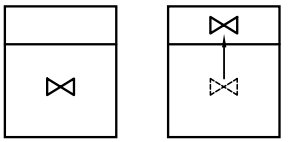
# Create Blocks Within a Drawing

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.

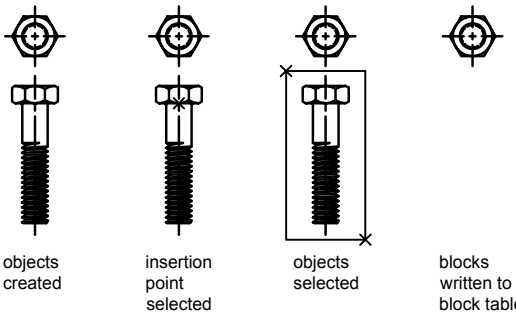
Each block definition includes a block name, one or more objects, the coordinate values of the base point to be used for inserting the block, and any associated attribute data.

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 452.




The illustration shows a typical sequence for creating a block definition within a drawing.




You can also use the Block Editor to create blocks that are saved within a drawing. For more information about using the Block Editor, see "Use the Block Editor" on page 463.

**To define a block for the current drawing**

1. Create the objects you want to use in the block definition.
2. On the Draw menu, click Block ➤ Make.
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.

Draw toolbar

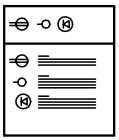
# Create Block Libraries

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 to define each block definition in the block library drawing, you can include a short description of the block that can be viewed in DesignCenter.

Optionally, you can also document each block definition by inserting it in the drawing area of the library drawing. In addition to the block geometry, you can include text that provides the block name, the date of creation, the date of the last modification, and any special instructions or conventions. This creates a visual index of the blocks in the block library drawing.



sample block
library drawing

Use DesignCenter to view and copy block definitions individually from block library drawings (or from any existing drawing) to your current drawing. DesignCenter does not overwrite an existing block definition in a drawing with one that comes from another drawing.

**To create a block library drawing**

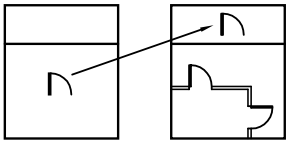
1. Begin a new drawing.
2. Define a block.
3. Repeat step 2 for as many related block definitions as you want to make.
4. Save the drawing using a name appropriate for a library drawing.

   These blocks can be inserted into any drawing using DesignCenter (ADCENTER).

Draw toolbar

# 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  or .
- Create and save only selected objects from your current drawing to a new drawing using  or .

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  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 (Xrefs)" on page 963.

### 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. On the Modify menu, click Object ➤ Block Description.
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.
5. Click OK.

# Use Tool Palettes to Organize Blocks

You can use tool palettes to organize blocks that are stored in one drawing file or separate drawing files.

Once you've added a block tool to a tool palette, you can easily insert the block reference in your drawing by dragging it from the tool palette to the drawing or by clicking and placing it in the drawing. For information about using tool palettes to organize and insert blocks, see "Tool Palettes" on page 22.

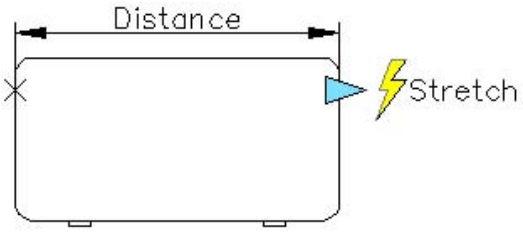
No procedures for this topic.

# Add Dynamic Behavior to Blocks

When you add dynamic behavior to a block definition, you add flexibility and intelligence to the block geometry. Instead of being a fixed part of a drawing, a dynamic block reference can be changed or manipulated as you work in a drawing.

## Quick Start to Creating Dynamic Blocks

You add dynamic behavior to new or existing block definitions by adding parameters and actions to the block in the Block Editor. In the following example, a desk block is shown in the Block Editor. The block contains a linear parameter, which displays similar to a dimension and is labeled "Distance," and a stretch action, which displays a lightning bolt and a "Stretch" label.




For a block to be dynamic, you must add at least one parameter. You then add an action and associate the action with the parameter. The types of parameters and actions you add to the block definition define how the block reference will work in a drawing. To see demonstrations of how parameters and actions are added to a block, see the New Features Workshop. Click Help ➤ New Features Workshop ➤ Produce ➤ Dynamic Blocks.

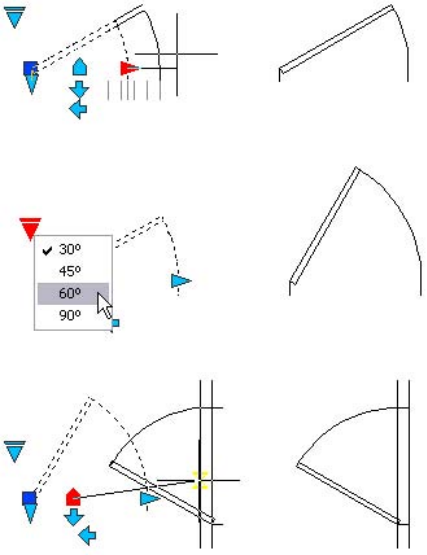
For more information in the Help system about creating dynamic blocks, see:

- "Overview of Dynamic Blocks" on page 460
- "Overview of Dynamic Block Elements" on page 474

# Overview of Dynamic Blocks

A *dynamic* block has flexibility and intelligence. A dynamic block reference can easily be changed in a drawing while you work. You can manipulate the geometry in a dynamic block reference through custom grips or custom properties. This allows you to adjust the block in-place as necessary rather than searching for another block to insert or redefining the existing one.

For example, if you insert a door block reference in a drawing, you might need to change the size of the door while you're editing the drawing. If the block is dynamic and defined to have an adjustable size, you can change the size of the door simply by dragging the custom grip or by specifying a different size in the Properties palette. You might also need to change the open angle of the door. The door block might also contain an alignment grip, which allows you to align the door block reference easily to other geometry in the drawing.




You use the Block Editor to create dynamic blocks. The Block Editor is a special authoring area where you add the elements that make a block dynamic. You can create a block from scratch, or you can add dynamic behavior to an existing block definition. You can also create geometry, just as you would in the drawing area.

You add *parameters* and *actions* to a block to make it dynamic. When you add these elements to the block, you add flexibility and intelligence to the block geometry.

- Parameters define custom properties for the dynamic block by specifying positions, distances, and angles for geometry in the block.
- Actions define how the geometry of a dynamic block reference will move or change when the block reference is manipulated in a drawing. When you add actions to the block, you must associate them with parameters and usually geometry.

When you add a parameter to a block definition, custom grips and properties are automatically added to the block. You use these custom grips and properties to manipulate the block reference in the drawing.

### Process for Creating Dynamic Blocks

To create quality dynamic blocks, so you get the results you expect, it is recommended that you follow the steps in the following process. This process will help you author dynamic blocks effectively.

#### Step 1. Plan the contents of the dynamic block before you create it

Before you create a dynamic block, you should know what it will look like and how it will be used in a drawing. Decide which objects within the block will change or move when the dynamic block reference is manipulated. Furthermore, decide *how* these objects will change. For example, you might create a dynamic block that can be resized. In addition, when the block reference is resized, additional geometry may be displayed. These factors determine the types of parameters and actions you add to the block definition, and how you make the parameters, actions, and geometry work together.

#### Step 2. Draw the geometry

You can draw the geometry for your dynamic block in the drawing area or the Block Editor. You can also use existing geometry in a drawing or an existing block definition.

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**NOTE** If you will use visibility states to change how geometry is displayed in the dynamic block references, you may not want to include all the geometry at this point. For more information about working with visibility states, see "Create Visibility States" on page 514.

---

### Step 3. Understand how the block elements will work together

Before you add parameters and actions to your block definition, understand their *dependencies* on each other and on the geometry within the block. When you add an action to the block definition, you will need to associate the action with a parameter and a selection set of geometry. This creates a dependency. When you add multiple parameters and actions to a dynamic block reference, you will need to set up the correct dependencies in order for the block reference to function properly in a drawing.

For example, you create a dynamic block that contains several objects. Some of the objects have a stretch action associated with them. You also want all the objects to rotate around the same base point. In this case, you should add the rotate action after adding all the other parameters and actions. If the rotate action isn't associated with all of the other objects (geometry, parameters, and actions) in the block definition, parts of the block reference may not rotate or manipulating the block reference may cause unexpected results.

### Step 4. Add parameters

Add the appropriate parameters to the dynamic block definition, following the prompts on the command line. For more information about using parameters, see "Use Parameters in Dynamic Blocks" on page 476.

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**NOTE** You can use the Parameter Sets tab of the Block Authoring Palettes to add a parameter and an associated action at the same time. For more information about using parameter sets, see "Use Parameter Sets" on page 508.

---

### Step 5. Add actions

Add the appropriate actions to the dynamic block definition. Follow the prompts on the command line, making sure to associate the actions with the correct parameters and geometry. For more information about using actions, see "Overview of Using Actions in Dynamic Blocks" on page 488.

### Step 6. Define how the dynamic block reference will be manipulated

You can specify how the dynamic block reference will be manipulated in the drawing. You can manipulate a dynamic block reference through custom grips and custom properties. When you create a dynamic block definition, you define what grips are displayed and how they edit the dynamic block reference. You also specify whether or not custom properties for the block will be shown in the Properties palette and whether or not these properties can be changed through the palette or through custom grips.

**Step 7. Save the block and then try it in a drawing**

Save the dynamic block definition and exit the Block Editor. Insert the dynamic block reference in a drawing and test the block's functionality.

**See also:**

"Save a Block in the Block Editor" on page 541

No procedures for this topic.

# Use the Block Editor

You use the Block Editor to add dynamic behavior to blocks. The Block Editor provides all the tools you need to add intelligence and flexibility to your blocks.

## Overview of the Block Editor

The Block Editor is a special authoring area for creating block definitions and adding dynamic behavior.

The Block Editor provides special Authoring palettes. These palettes provide quick access to block authoring tools.

In addition to the Block Authoring palettes, the Block Editor provides a drawing area in which you can draw and edit geometry as you would in the program's main drawing area. You can specify the background color for the Block Editor drawing area.

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**NOTE** You can use most commands in the Block Editor. When you enter a command that is not allowed in the Block Editor, a message is displayed on the command line.

---

You can use the Block Editor to edit or add dynamic behavior to block definitions that exist in the current drawing. You can also use it to create new block definitions.

In the Block Editor, a special toolbar is displayed above the drawing area. The toolbar shows the name of the block definition currently being edited and provides tools to do the following:

- Save the block definition
- Add a parameter
- Add an action
- Define attributes

- Close the Block Editor
- Manage visibility states

You can select any parameter, grip, action, or geometric object in the Block Editor to view its properties in the Properties palette. When you select an object in the Block Editor, the coordinate values shown in the Properties palette reflect the block definition space.

When you work in the Block Editor, the command line should be displayed. The command line displays prompts for nearly all aspects of creating dynamic blocks.

### UCS in the Block Editor

A UCS icon is displayed in the drawing area of the Block Editor. The origin of the UCS icon defines the base point for the block. You can change the base point for the block by moving the geometry relative to the origin of the UCS icon, or by adding a base point parameter.

The UCS command is disabled in the Block Editor. You can open an existing 3D block definition in the Block Editor and assign parameters to the block. However, the parameters will ignore any *Z* coordinate values in the block space. Consequently, the block reference cannot be edited along the *Z* axis. Furthermore, while you can create a dynamic block that contains solid objects and add actions to it, such as move, rotate, and scale, you can't perform solid editing features within a dynamic block reference (for example, stretch a solid, move a hole within a solid, and so on).

**See also:**

"Define Block Attributes" on page 552

### To open an existing block definition in the Block Editor

1. Click Tools menu ➤ Block Editor.
2. In the Edit Block Definition dialog box, do one of the following:
   - Select a block definition from the list.
   - Select <Current Drawing> if the drawing is the block definition you want to open.
3. Click OK.

Standard toolbar

Right-click a selected block. Click Block Editor.

**To create a new block definition in the Block Editor**

1. Click Tools menu ➤ Block Editor.
2. In the Edit Block Definition dialog box, enter a name for the new block definition.
3. Click OK.
4. In the Block Editor, click Save Block Definition.

   **NOTE** This saves the block definition even if you have not added any objects in the Block Editor drawing area.

5. Click Close Block Editor.

Standard toolbar

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

1. If the Tool Palettes window is not already open, click Tools menu ➤ Tool Palettes Window.
2. Right-click a block icon.
3. Click Block Editor.

**NOTE** A block on a tool palette may reside in another drawing. The drawing that contains the block definition is opened in the Block Editor.

Standard toolbar

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

1. If DesignCenter is not already open, click Tools menu ➤ DesignCenter.

2 Right-click a block icon.

3 Click Block Editor.

Standard toolbar

**To open a drawing file saved as a block (not dynamic) in the Block Editor**

1 Open the drawing file that is saved as a block.

2 Click Tools menu ➤ Block Editor.

3 In the Edit Block Definition dialog box, select <Current Drawing>.

4 Click OK.

Standard toolbar

**To open a drawing file saved as a dynamic block in the Block Editor**

1 Open the drawing file that is saved as a block.

An alert is displayed that states that the drawing file contains authoring elements.

2 In the alert dialog box, click Yes to open the drawing in the Block Editor.

Standard toolbar

**To view properties of a block definition in the Block Editor**

1 Click Tools menu ➤ Block Editor.

2 In the Edit Block Definition dialog box, do one of the following:

- Select a block definition from the list.
- Select <Current Drawing> if the drawing is the block definition you want to open.

3 Click OK.

4 Make sure the Properties Palette window is displayed. If it's not displayed, click Tools menu ➤ Properties.

5 In the Properties Palette window, under Block, view the properties of the block definition.

Standard toolbar

**To view properties of objects in the Block Editor**

1 In the Block Editor, select an object.

2 Make sure the Properties Palette window is displayed. If it's not displayed, click Tools menu ➤ Properties.

3 In the Properties Palette window, view the properties of the selected object.

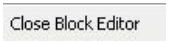
Standard toolbar

Right-click the selected object. Click Properties.

**To close the Block Editor**

■ In the Block Editor, click Close Block Editor.

Block Editor toolbar




**To prevent access to the Block Editor**

■ On the command line, enter **blockedtlock**.

■ Enter **1**, and then press ENTER.

# Use Block Authoring Palettes

The Block Editor has three block authoring palettes: Parameters, Actions, and Parameter Sets.

The Block Authoring Palettes window is displayed only in the Block Editor. Use these palettes to add parameters and actions to your dynamic block definition.

### Create Custom Block Authoring Tools

You can create custom block authoring tools. In order to preserve the default tools on the block authoring palettes, you should create a new palette for custom block authoring tools. You can then copy a parameter set tool from one of the existing palettes and paste the copy onto the new palette. In the Tool Properties Dialog Box, you can then change the properties of the new tool, including the following:

- Tool description
- Parameter type
- Associated action(s)
- Key point on the parameter to which the action is tied (if applicable)
- Tool palette image

You cannot drag parameters and actions from the Block Editor onto any tool palette.

**See also:**


"Use Parameters in Dynamic Blocks" on page 476
"Use Actions with Dynamic Blocks" on page 488
"Use Parameter Sets" on page 508
"Control Tool Properties" on page 31
"Organize Tool Palettes" on page 38


### To show or hide the Block Authoring palettes in the Block Editor

- In the Block Editor, on the Block Editor toolbar, click the Block Authoring Palettes button.

Block Editor toolbar

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**To make a copy of a parameter set**

1. In the Block Editor, in the Block Authoring Palettes window, Parameters Sets tab, right-click a parameter set. Click Copy.
2. Right-click anywhere on the palette to which you want to add the parameter set (except on a parameter set). Click Paste.

**To add an action to a parameter set**

1. In the Block Editor, in the Block Authoring Palettes window, Parameters Sets tab, right-click a parameter set. Click Properties.
2. In the Tool Properties dialog box, under Parameter, click Actions, and then click the [...] button.
3. In the Add Actions dialog box, under Action Object to Add, select an action from the list.
4. Click Add.
5. (Optional) Repeat steps 3 and 4 to add additional actions.
6. Click OK.
7. In the Tool Properties dialog box, click OK.

**To delete an action from a parameter or parameter set**

1. In the Block Editor, in the Block Authoring Palettes window, right-click a parameter or parameter set. Click Properties.
2. In the Tool Properties dialog box, under Parameter, click Actions, and then click the [...] button.
3. In the Add Actions dialog box, under Action Object List, select an action from the list.
4. Click Delete.
5. (Optional) Repeat steps 3 and 4 to delete additional actions.
6. Click OK.
7. In the Tool Properties dialog box, click OK.

## How Objects Are Displayed in the Block Editor

Parameters, actions, and their relationships (dependencies) are displayed in different ways in the Block Editor. You can specify settings for some of these elements.

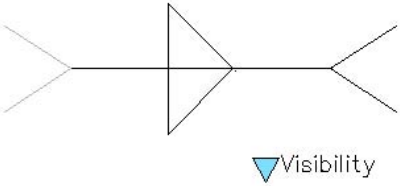
### Parameters

In the Block Editor, most parameters look like dimensions. If you create a value set (a range or list of values) for a parameter, tick marks are shown at the locations of those values.

You can specify the following settings for parameters in the Block Editor:

- Parameter color
- Parameter text and arrow size
- Parameter font
- Grip color
- Display of value set markers (tick marks) for parameters

When you use a visibility parameter in your dynamic block definition, you can specify which geometric objects are invisible for a given visibility state. You can specify whether or not geometry that is made invisible for visibility states displays in the Block Editor. In the following example, a visibility state is displayed in the Block Editor. The geometry that displays in a dimmed state is invisible for that visibility state.




### Actions

An action displays its name and icon (a lightning bolt) in the Block Editor. You can specify the text size and color for actions in the Block Editor.

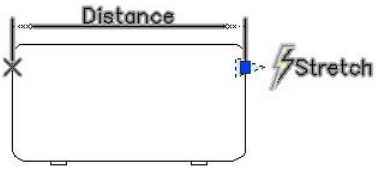
### Grips

You can specify grip size and color for display in the Block Editor. This setting does not affect the size and color of the grips in a block reference in a drawing.

### Dependencies

When you select a parameter, grip, or action in the Block Editor, its associated objects, or dependencies, are highlighted. This is called *dependency highlighting*. You can turn dependency highlighting on or off.

The following example shows how dependency highlighting creates a halo effect for the associated parameter (labeled Distance) and action (labeled Stretch) when you select the custom grip in the Block Editor.




The following table details what is dependency highlighted when you select an element in the Block Editor.

| Selected object in the Block Editor | Objects that are dependency highlighted |
|---|---|
| Parameter | Associated grips and actions |
| Grip | Associated parameter and actions |
| Action | Associated parameters, grips, and the selection set (geometry) |

#### To specify the display color for parameters in the Block Editor

1. On the command line, enter **bparametercolor**.
2. Enter one of the following values:
   - **BYLAYER**
   - **BYBLOCK**
   - An integer from 1 to 255
   - A true color specified by three integers each ranging from 1 to 255 in the following format: **RGB:000,000,000**.
3. Press ENTER.

#### To specify the text size for parameters and actions in the Block Editor

1. On the command line, enter **bparametersize**.
2. Enter an integer from 1 to 255 (pixels).

3 Press ENTER.

**To specify the font for parameters in the Block Editor**

1 On the command line, enter **bparameterfont**.

2 Enter any TrueType or SHX font on your system.

3 Press ENTER.

**To specify the text color for actions in the Block Editor**

1 On the command line, enter **bactioncolor**.

2 Enter one of the following values:

- **BYLAYER**
- **BYBLOCK**
- An integer from 1 to 255.
- A true color specified by three integers each ranging from 1 to 255 in the following format: **RGB:000,000,000**.

3 Press ENTER.

**To specify the display size of grips in the Block Editor**

1 On the command line, enter **bgripobjsize**.

2 Enter an integer from 1 to 255 (pixels).

3 Press ENTER.

**To specify the display color of grips in the Block Editor**

1 On the command line, enter **bgripobjcolor**.

2 Enter one of the following values:

- **BYLAYER**
- **BYBLOCK**
- An integer from 1 to 255.
- A true color specified by three integers each ranging from 1 to 255 in the following format: **RGB:000,000,000**.

3 Press ENTER.

**To update the text and grip display size at their specified values in the Block Editor**

1 In the Block Editor, on the Block Editor toolbar, click Update Parameter and Action Text Size.

---

**NOTE** When you zoom in and out in the Block Editor, text and grip size will change relative to the zoom factor. However, you can update the Block Editor to display the text and grip size at their specified values.

---

**To specify whether or not objects that are invisible for a visibility state can be seen in the Block Editor**

1 On the command line, enter **bvmode**.

2 Enter one of the following values:

- **0** Specifies that objects that are invisible are not visible in the Block Editor.
- **1** Specifies that objects that are invisible are dimmed in the Block Editor.

3 Press ENTER.

Block Editor toolbar

**To turn dependency highlighting on or off in the Block Editor**

1 On the command line, enter **bdependencyhighlight**.

2 Enter one of the following values:

- **0** Turns off dependency highlighting.
- **1** Turns on dependency highlighting.

3 Press ENTER.

**To control whether value set markers (tick marks) are displayed**

1 On the command line, enter **btmarkdisplay**.

2 Enter one of the following values:

- **0** Specifies that value set markers are not displayed.
- **1** Specifies that value set markers are displayed.

3 Press ENTER.

# Add Dynamic Elements to Blocks

To make a block dynamic, you need to add specific elements to the block definition.

## Overview of Dynamic Block Elements

You add dynamic elements to a block definition in the Block Editor. In addition to geometry, a dynamic block generally includes one or more parameters and one or more actions.

- **Parameters.** Define custom properties for the dynamic block by specifying positions, distances, and angles for geometry in the block.
- **Actions.** Define how the geometry of a dynamic block reference will move or change when you manipulate the block reference in a drawing. When you add actions to a dynamic block definition, you must associate the actions with parameters. You also specify a selection set of geometry that the action will affect.

---

**NOTE** Parameters and actions are displayed only in the Block Editor. When you insert a dynamic block reference in a drawing, the parameters and actions contained in the dynamic block definition are not displayed.

---

When you add a parameter to a dynamic block definition, grips are added to key points of the parameter. *Key points* are the parts of a parameter that you use to manipulate the block reference. For example, a linear parameter has key points at its base point and end point. You can manipulate the parameter distance from either key point.

The type of parameter that you add to a dynamic block determines the type of grips that are added. Each type of parameter supports only certain types of actions. The following table shows the relationship among parameters, grips, and actions.

| Parameter Type | Grip Type | Actions You Can Associate with a Parameter |
|---|---|---|
| Point | Standard | Move, Stretch |
| Linear | Linear | Move, Scale, Stretch, Array |
| Polar | Standard | Move, Scale, Stretch, Polar Stretch, Array, |
| XY | Standard | Move, Scale, Stretch, Array |
| Rotation | Rotation | Rotate |
| Flip | Flip | Flip |
| Alignment | Alignment | None (The action is implied and contained within the parameter.) |
| Visibility | Lookup | None (The action is implied and controlled by visibility states.) |
| Lookup | Lookup | Lookup |
| Base | Standard | None |

**To create a dynamic block**

1. Click Tools menu ➤ Block Editor.
2. In the Edit Block Definition dialog box, do one of the following:
   - Select a block definition from the list.
   - Select <Current Drawing> if you want to save the drawing as a dynamic block.
   - Under Block to Create or Edit, enter a name for a new block definition.

3 Click OK.

4 In the Block Editor, add or edit geometry as necessary.

5 Do one of the following:

- Add one or more parameter sets from the Parameter Sets tab of the Block Authoring Palettes, following the prompts on the command line. Double-click the yellow alert icon (or use the  command) and follow the prompts on the command line to associate the action with a selection set of geometry.
- Add one or more parameters from the Parameters tab of the Block Authoring Palettes, following the prompts on the command line. Add one or more actions from the Actions tab, following the prompts on the command line.

6 On the Block Editor toolbar, click Save Block Definition.

7 Click Close Block Editor.

Standard toolbar

Right-click a block. Click Block Editor.

## Use Parameters in Dynamic Blocks

You add parameters to a dynamic block definition in the Block Editor. In the Block Editor, parameters have an appearance similar to dimensions. Parameters define custom properties for the block. Parameters also specify positions, distances, and angles for geometry in the block reference. When you add a parameter to a dynamic block definition, the parameter defines one or more custom properties for the block.

For example, when you add a rotation parameter to a dynamic block definition, the rotation parameter defines an *Angle* property for the block reference. So, if you have a block of a chair in your drawing, and you want to be able to rotate the block's position as you edit, the parameter defines the axis whereby the block can be rotated.

If you add a point parameter to a dynamic block definition, the point parameter defines two custom properties for the block reference: *Position X* and *Position Y* (relative to the base point of the block reference.)

A dynamic block definition must contain at least one parameter. When a parameter is added to a dynamic block definition, grips associated with key

points of the parameter are automatically added. You must then add an action to the block definition and associate the action with a parameter.

In a drawing, you use a grip or custom property in the Properties palette to manipulate the block reference. When you manipulate the block reference in a drawing, by moving a grip or changing the value of a custom property in the Properties palette, you change the value of the parameter that defines that custom property in the block. When you change the value of the parameter, it drives the action that is associated with that parameter, which changes the geometry (or a property) of the dynamic block reference.

Parameters also define and constrain values that affect the dynamic block reference's behavior in a drawing. Some parameters can have a fixed set of values, minimum and maximum values, or increment values. For example, a linear parameter used in a window block may have the following fixed set of values: 10, 20, 30, and 40. When the block reference is inserted in a drawing, you can only change the window to one of these values. Adding a value set to a parameter allows you to limit how the block reference is manipulated in a drawing. For more information about parameter value sets, see "Specify Value Sets for Dynamic Blocks" on page 532.

The point, linear, polar, XY, and rotation parameters have a property called Chain Actions. For more information about allowing chained actions for a parameter, see "Allow Chained Actions for Dynamic Blocks" on page 536.

You can also extract parameter values through the Attribute Extraction wizard or through an attribute extraction template file.

The following table lists and describes the types of parameters you can add to a dynamic block definition and the types of actions you can associate with each parameter.

| Parameter Type | Description | Supported Actions |
|---|---|---|
| Point | Defines an *X* and *Y* location in the drawing. In the Block Editor, looks similar to an ordinate dimension. | Move, Stretch |
| Linear | Shows the distance between two anchor points. Constrains grip movement along a preset angle. In the Block Editor, looks similar to an aligned dimension. | Move, Scale, Stretch, Array |
| Polar | Shows the distance between two anchor points and displays an angle value. You can use both grips and the Properties palette to change both the distance value and the angle. In the Block Editor, looks similar to an aligned dimension | Move, Scale, Stretch, Polar Stretch, Array, |
| XY | Shows the *X* and *Y* distances from the base point of the parameter. In the Block Editor, displays as a pair of dimensions (horizontal and vertical). | Move, Scale, Stretch, Array |

| Parameter Type | Description | Supported Actions |
|---|---|---|
| Rotation | Defines an angle. In the Block Editor, displays as a circle. | Rotate |
| Flip | Flips objects. In the Block Editor, displays as a reflection line. Objects can be flipped about this reflection line. Displays a value that shows if the block reference has been flipped or not. | Flip |
| Alignment | Defines an *X* and *Y* location and an angle. An alignment parameter always applies to the entire block and needs no action associated with it. An alignment parameter allows the block reference to automatically rotate around a point to align with another object in the drawing. An alignment parameter affects the rotation property of the block reference. In the Block Editor, looks like an alignment line. | None (The action is implied and contained within parameter.) |
| Visibility | Controls the visibility of objects in the block. A visibility parameter always applies to the entire block and needs no action associated with it. In a drawing, you click the grip to display a list of visibility states available for the block reference. In the Block Editor, displays as text with an associated grip. | None (The action is implied and controlled by visibility states.) |
| Lookup | Defines a custom property that you can specify or set to evaluate a value from a list or table you define. It can be associated with a single lookup grip. In the block reference, you click the grip to display a list of available values. In the Block Editor, displays as text with an associated grip. | Lookup |
| Base | Defines a base point for the dynamic block reference relative to the geometry in the block. Cannot be associated with any actions, but can belong to an action's selection set. In the Block Editor, displays as a circle with crosshairs. | None |

**See also:**


"Specify Value Sets for Dynamic Blocks" on page 532
"Overview of Specifying Custom Properties for Dynamic Blocks" on page 529
"Extract Data from Block Attributes" on page 554
"Extract Block Attribute Data (Advanced)" on page 555


### To add a point parameter to a dynamic block definition

1. In the Block Editor, in the Block Authoring Palettes window, Parameters tab, click the Point Parameter tool.

2 (Optional) Follow the prompts on the command line to specify the following for the parameter:

- Name
- Label
- Description
- Chained Actions
- Properties displayed for block reference

**NOTE** You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.

3 Do one of the following to specify a location for the parameter:

- Use the pointing device.
- Enter the *X* and *Y* coordinate values, separated by a comma.

4 Do one of the following to specify a location for the parameter label:

- Use the pointing device.
- Enter the *X* and *Y* coordinate values, separated by a comma.

  The yellow alert icon indicates that you should associate an action with the parameter you just added.

5 To add an action now, double-click the alert icon. Follow the prompts to associate an action with the parameter and a selection set of geometry.

6 On the Block Editor toolbar, click Save Block Definition.

7 If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar

**To add a linear parameter to a dynamic block definition**

1 In the Block Editor, in the Block Authoring Palettes window, Parameters tab, click the Linear Parameter tool.

2 (Optional) Follow the prompts on the command line to specify the following for the parameter:

- Name

- Label
- Description
- Value set
- Chained Actions
- Properties displayed for block reference

**NOTE** You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.

3. Do one of the following to specify a base point for the parameter:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.
4. Do one of the following to specify an endpoint for the parameter:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.
5. Do one of the following to specify a location for the parameter label:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.

   The yellow alert icon indicates that you should associate an action with the parameter you just added.
6. To add an action now, double-click the alert icon. Follow the prompts to associate an action with the parameter and a selection set of geometry.
7. On the Block Editor toolbar, click Save Block Definition.
8. If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar

**To add a polar parameter to a dynamic block definition**

1. In the Block Editor, in the Block Authoring Palettes window, Parameters tab, click the Polar Parameter tool.

2. (Optional) Follow the prompts on the command line to specify the following for the parameter:
   - Name
   - Label
   - Description
   - Value set
   - Chained Actions
   - Properties displayed for block reference

   **NOTE** You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.

3. Do one of the following to specify a base point for the parameter:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.
4. Do one of the following to specify an endpoint for the parameter:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.
5. Do one of the following to specify a location for the parameter label:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.

   The yellow alert icon indicates that you should associate an action with the parameter you just added.
6. To add an action now, double-click the alert icon. Follow the prompts to associate an action with the parameter and a selection set of geometry.
7. On the Block Editor toolbar, click Save Block Definition.
8. If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar

**To add an XY parameter to a dynamic block definition**

1. In the Block Editor, in the Block Authoring Palettes window, Parameters tab, click the XY Parameter tool.
2. (Optional) Follow the prompts on the command line to specify the following for the parameter:
   - Name
   - Label
   - Description
   - Value set
   - Chained Actions
   - Properties displayed for block reference

   **NOTE** You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.
3. Do one of the following to specify a base point for the parameter:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.
4. Do one of the following to specify an endpoint for the parameter:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.

   The yellow alert icon indicates that you should associate an action with the parameter you just added.
5. To add an action now, double-click the alert icon. Follow the prompts to associate an action with the parameter and a selection set of geometry.
6. On the Block Editor toolbar, click Save Block Definition.
7. If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar

### To add a rotation parameter to a dynamic block definition

1. In the Block Editor, in the Block Authoring Palettes window, Parameters tab, click the Rotation Parameter tool.
2. (Optional) Follow the prompts on the command line to specify the following for the parameter:
   - Name
   - Label
   - Description
   - Value set
   - Chained Actions
   - Properties displayed for block reference

   **NOTE** You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.
3. Do one of the following to specify a base point for the parameter:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.
4. Do one of the following to specify the radius for the parameter:
   - Use the pointing device.
   - Enter a value.
5. Specify a base angle for the parameter.

   The angle shown in the Properties palette when you select the block reference in a drawing is measured relative to the base angle specified in the block definition.
6. Do one of the following to specify a default rotation angle for the parameter:
   - Use the pointing device.
   - Enter an angle value.
7. Do one of the following to specify a location for the parameter label:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.

The yellow alert icon indicates that you should associate an action with the parameter you just added.

8. To add an action now, double-click the alert icon. Follow the prompts to associate an action with the parameter and a selection set of geometry.
9. On the Block Editor toolbar, click Save Block Definition.
10. If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar

**To add an alignment parameter to a dynamic block definition**

1. In the Block Editor, in the Block Authoring Palettes window, Parameters tab, click the Alignment Parameter tool.
2. Follow the prompts on the command line to specify the parameter name.
3. Do one of the following to specify the base point of the alignment parameter:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.

   This base point is the default location for the alignment parameter grip.
4. (Optional) Enter **type** on the command line and follow the prompts to specify whether the parameter type is perpendicular or tangent.
5. Do one of the following to specify the alignment direction:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.
6. On the Block Editor toolbar, click Save Block Definition.
7. If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar

**To add a flip parameter to a dynamic block definition**

1. In the Block Editor, in the Block Authoring Palettes window, Parameters tab, click the Flip Parameter tool.
2. (Optional) Follow the prompts on the command line to specify the following for the parameter:
   - Name
   - Label
   - Description
   - Properties displayed for block reference

   **NOTE** You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.
3. Do one of the following to specify the first point of the reflection line for the flip parameter:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.

   This first point of the reflection line is the default location for the flip parameter grip.
4. Do one of the following to specify the second point of the reflection line for the flip parameter:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.
5. Do one of the following to specify a location for the parameter label:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.

   The yellow alert icon indicates that you should associate an action with the parameter you just added.
6. To add an action now, double-click the alert icon. Follow the prompts to associate an action with the parameter and a selection set of geometry.
7. On the Block Editor toolbar, click Save Block Definition.
8. If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar

**To add a visibility parameter to a dynamic block definition**

1. In the Block Editor, in the Block Authoring Palettes window, Parameters tab, click the Visibility Parameter tool.
2. (Optional) Follow the prompts on the command line to specify the following for the parameter:
   - Name
   - Label
   - Description
   - Properties displayed for block reference

   **NOTE** You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.
3. Do one of the following to specify a location for the parameter:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.

   This is the location for the visibility parameter grip in the block reference.
4. On the Block Editor toolbar, click Save Block Definition.
5. If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar

**To add a lookup parameter to a dynamic block definition**

1. In the Block Editor, in the Block Authoring Palettes window, Parameters tab, click the Lookup Parameter tool.
2. (Optional) Follow the prompts on the command line to specify the following for the parameter:
   - Name
   - Label

- Description
- Properties displayed for block reference

**NOTE** You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.

3. Do one of the following to specify a location for the parameter:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.

   The yellow alert icon indicates that you should associate an action with the parameter you just added.
4. To add an action now, double-click the alert icon. Follow the prompts to associate a lookup action with the parameter. This will display the Property Lookup Table dialog box, which you can complete now or later.
5. Click OK.
6. On the Block Editor toolbar, click Save Block Definition.
7. If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar

**To add a base point parameter to a dynamic block definition**

1. In the Block Editor, in the Block Authoring Palettes window, Parameters tab, click the Base Point Parameter tool.
2. Do one of the following to specify a location for the parameter:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.
3. On the Block Editor toolbar, click Save Block Definition.
4. If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar

# Use Actions with Dynamic Blocks

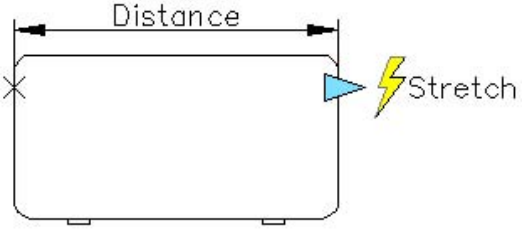
Actions define how the geometry of a dynamic block reference will move or change when the custom properties of the block reference are manipulated in a drawing.

## Overview of Using Actions in Dynamic Blocks

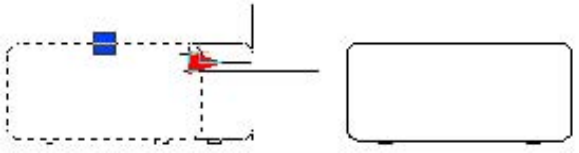
Actions define how the geometry of a dynamic block reference will move or change when the custom properties of a block reference are manipulated in a drawing. A dynamic block usually contains at least one action.

In general, when you add an action to a dynamic block definition, you must associate the action with a parameter, a key point on the parameter, and geometry. A *key point* is the point on a parameter that drives its associated action when edited. The geometry associated with an action is called the *selection set*.

In the following example, the dynamic block definition contains geometry that represents a desk, a linear parameter with one grip specified for its endpoint, and a stretch action associated with the endpoint of the parameter and the geometry for the right side of the desk. The endpoint of the parameter is the key point. The geometry on the right side of the desk is the selection set.




When you want to change the block reference in a drawing, you move the grip, and the desk stretches.



The types of actions you can use in a dynamic block are

- Move

- Scale
- Stretch
- Polar Stretch
- Rotate
- Flip
- Array
- Lookup

You can assign more than one action to the same parameter and geometry. However, you should not assign two or more of the same type of actions to the same key point on a parameter if both actions affect the same geometry. This can result in unexpected behavior in the block reference.

#### Actions and Parameter Types

Each type of action can be associated with specific parameters. The following table shows the parameters with which you can associate each type of action.

| Action Type | Parameter |
|---|---|
| Move | Point, linear, polar, XY |
| Scale | Linear, polar, XY |
| Stretch | Point, linear, polar, XY |
| Polar Stretch | Polar |
| Rotate | Rotation |
| Flip | Flip |
| Array | Linear, polar, XY |
| Lookup | Lookup |

## Use a Move Action in a Dynamic Block

The move action is similar to the  command. In a dynamic block reference, a move action causes objects to move a specified distance and angle.

In a dynamic block definition, you can associate a move action with any of the following parameters:

- Point

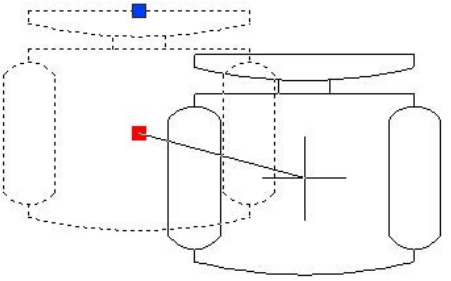
- Linear
- Polar
- XY

After associating a move action with a parameter, you associate the action with a selection set of geometry.

In a dynamic block reference, if you change the value of a parameter associated with a move action through a grip or the Properties palette, it may affect the key point on the parameter associated with the move action. If the key point is affected, the geometry in the move action's selection set will move.

For example, you have a dynamic block that represents a chair. The block contains a point parameter and a move action associated with the point parameter. The move action's selection set contains all the geometry in the block (the chair). When you use the grip associated with the point parameter (or the Position X or Position Y properties in the Properties palette) to manipulate the dynamic block reference, it changes the value of the point parameter. This change in value causes the chair to move.

In the following example, when the chair block reference is moved by dragging the point parameter grip, the new position of this grip is reported in the Properties palette



### Specify the Distance Type Property for a Move Action

When a move action is associated with an XY parameter, the move action has an override property called *Distance Type*. This property specifies whether the distance applied to the move is the parameter's *X* value, *Y* value, or *X* and *Y* coordinate value from the parameter's base point.

For example, you specify X Distance as the Distance Type for a move action in a dynamic block definition. That means that the block can only be moved on the *X* axis. So if you try to move the block on the *Y* axis, it doesn't move.

**See also:**

"Use Distance Multiplier and Angle Offset Action Overrides" on page 507

### To add a move action to a dynamic block definition

1. In the Block Editor, in the Block Authoring Palettes window, Actions tab, click the Move Action tool.
2. In the Block Editor drawing area, select a parameter to associate with the action. (You can associate the following parameter types with a move action: point, linear, polar, and XY.)
3. Select the objects (the selection set) to associate with the action.
4. Press ENTER.
5. (Optional) Follow the prompts on the command line to specify a distance multiplier and angle offset.
6. Do one of the following to specify the location of the action:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.

   **NOTE** The action location in the block definition has no effect on the appearance or functionality of the block reference.
7. On the Block Editor toolbar, click Save Block Definition.
8. If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar

### To specify the Distance Type property for a move action associated with an XY parameter

1. In the Block Editor, select a move action.
2. On the Properties palette, Overrides area, Distance Type, select an option from the list.

## Use a Scale Action in a Dynamic Block

The scale action is similar to the  command. In a dynamic block reference, a scale action causes the block's selection set to scale when the associated parameter is edited by moving grips or by using the Properties palette.

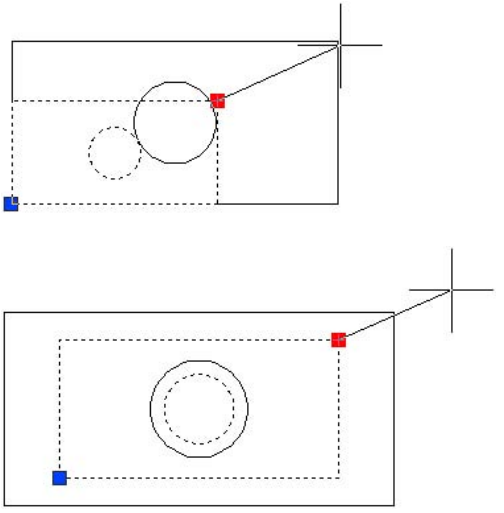
In a dynamic block definition, you associate a scale action with an entire parameter, not a key point on the parameter. You can associate a scale action with any of the following parameters:

- Linear
- Polar
- XY

After associating a scale action with a parameter, you associate the action with a selection set of geometry.

### Specify the Type of Base Point for a Scale Action

The scale action has a property called *Base Type*. With this property, you specify whether the base point for the scale factor is *dependent* or *independent*.



### Specify the Scale Type Property for a Scale Action

When a scale action is associated with an XY parameter, the scale action has an override property called *Scale Type*. This property specifies whether the

scale factor applied is the parameter's *X* distance, *Y* distance, or *X* and *Y* coordinate value distance from the parameter's base point.

For example, you specify X Distance as the Scale Type for a scale action in a dynamic block definition. In a drawing, when you edit the block reference by dragging a grip on the XY parameter only along the Y axis, the associated geometry does not scale.

**To add a scale action to a dynamic block definition**

1. In the Block Editor, in the Block Authoring Palettes window, Actions tab, click the Scale Action tool.
2. In the Block Editor drawing area, select a parameter to associate with the action. (You can associate the following parameter types with a scale action: linear, polar, and XY.)
3. Select the geometry (the selection set) to associate with the action.
4. Press ENTER.
5. (Optional) Follow the prompts on the command line to specify the type of base point.
6. Do one of the following to specify the location of the action:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.

   **NOTE** The action location in the block definition has no effect on the appearance or functionality of the block reference.
7. On the Block Editor toolbar, click Save Block Definition.
8. If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar

**To specify an independent base point for a scale action in a dynamic block definition**

1. In the Block Editor, select a scale action.
2. On the Properties palette, Overrides area, Base Type, select Independent from the drop-down list.

**To specify the Scale Type property for a scale action associated with an XY parameter**

1. In the Block Editor, select a scale action associated with an XY parameter.
2. On the Properties palette, Overrides area, Scale Type, select an option from the drop-down list.

   Depending on the override you apply, the scale action for the block is limited to the *X* axis, the *Y* axis, or both.

## Use a Stretch Action in a Dynamic Block

In a dynamic block reference, a stretch action causes objects to move and stretch a specified distance in a specified location.

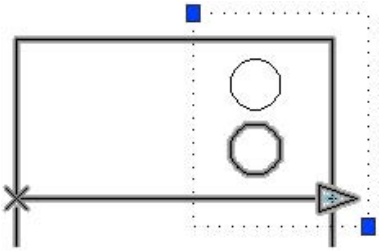
In a dynamic block definition, you can associate a stretch action with any of the following parameters:

- Point
- Linear
- Polar
- XY

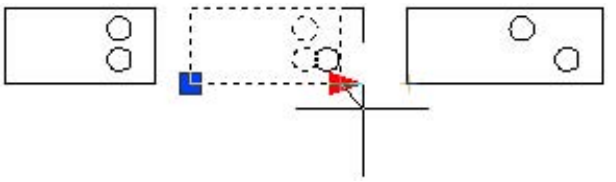
After associating a stretch action with a parameter, you specify a stretch frame for the stretch action. You then select the objects for the stretch action's selection set. The stretch frame determines how the objects within or crossed by the frame are edited in the block reference. The behavior is similar to specifying a crossing selection window with the  command.

- Objects entirely within the frame are moved.
- Objects that cross the frame are stretched.
- Objects within or crossed by the frame, but not included in the selection set, are not stretched or moved.
- Objects outside the frame and included in the selection set are moved.

In the following example, the stretch frame is indicated by the dashed line, and the selection set has a haloed effect. The top circle, while enclosed by the stretch frame, is not included in the selection set, so the top circle won't move. The bottom circle is entirely enclosed in the stretch frame and included in the selection set, so the bottom circle will move. The rectangle is crossed by the stretch frame and included in the selection set, so the rectangle will stretch.



In a dynamic block reference, if you change the value of a parameter associated with a stretch action through a grip or the Properties palette, it may affect the key point on the parameter associated with the stretch action. If the key point is affected, the geometry in the stretch action's selection set will move.



### Specify the Distance Type Property for a Stretch Action

When a stretch action is associated with an XY parameter, the stretch action has an override property called *Distance Type*. This property specifies whether the distance applied to the move is the parameter's *X* value, *Y* value, or *X* an *Y* coordinate value from the parameter's base point.

For example, you specify X Distance as the Distance Type for a stretch action in a dynamic block definition. In a drawing, when you try to edit the block reference by dragging the key point only along the *Y* axis, the associated geometry does not move because you've added the Distance Type override allowing only a move on the *X* axis.

**See also:**

"Use Distance Multiplier and Angle Offset Action Overrides" on page 507

### To add a stretch action to a block definition

1. In the Block Editor, in the Block Authoring Palettes window, Actions tab, click the Stretch Action tool.
2. In the Block Editor drawing area, select a parameter to associate with the action. (You can associate the following parameter types with a move action: point, linear, polar, and XY.)

If you associated the stretch action with a point parameter, go to step 4.

3. Do one of the following to select a parameter point to associate with the action:
   - Use the pointing device.
   - Follow the prompts on the command line.
4. Do one of the following to specify the first corner of the stretch frame:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.
5. Do one of the following to specify the opposite corner of the stretch frame:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.
6. Select the objects for the selection set.
7. Press ENTER.
8. (Optional) Follow the prompts on the command line to specify a distance multiplier and angle offset.
9. Do one of the following to specify the location of the action:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.

   **NOTE** The action location in the block definition has no effect on the appearance or the functionality of the block reference.
10. On the Block Editor toolbar, click Save Block Definition.
11. If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar

**To specify the Distance Type property for a stretch action associated with an XY parameter**

1. In the Block Editor, select a stretch action associated with an XY parameter.
2. On the Properties palette, Overrides area, Distance Type, select an option from the drop-down list.

Depending on the override you apply, the stretch action for the block is limited to the *X* axis, the *Y* axis, or both.

## Use a Polar Stretch Action in a Dynamic Block

In a block reference, a polar stretch action rotates, moves, and stretches objects a specified angle and distance when the key point on the associated polar parameter is changed through a grip or the Properties palette.

In a dynamic block definition, you can only associate a polar stretch action with a polar parameter. The base point for the stretch part of the action is the parameter point opposite the key point.

After associating a polar stretch action with a polar parameter, you specify a stretch frame for the polar stretch action. You then select objects to stretch and objects to rotate.

Objects entirely within the frame are moved.

Objects that cross the frame are stretched.

Objects in the action's selection set specified to rotate only are not stretched.

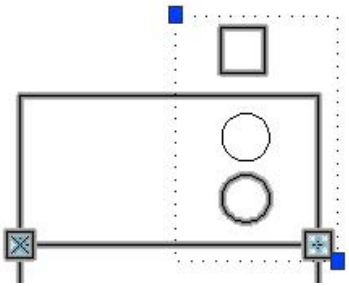
Objects within the frame are moved linearly after they are rotated.

Objects crossed by the frame are stretched linearly after they are rotated.

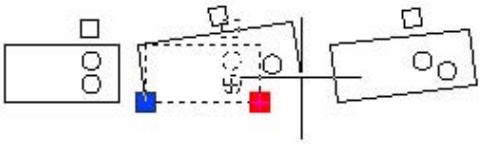
Objects within or crossed by the frame, but not included in the selection set, are not stretched or rotated.

- Objects outside the frame and included in the selection set are moved.

In the following example, the stretch frame is indicated by the dashed line, and the selection set has a haloed effect. The top circle, while enclosed by the stretch frame, is not included in the selection set, so the top circle won't move. The bottom circle is entirely enclosed in the stretch frame and included in the stretch selection set, so the bottom circle will move. The rectangle is crossed by the stretch frame and included in the selection set, so the rectangle will stretch. The square is entirely enclosed in the stretch frame and included in the rotate selection set, but not the stretch selection set, so the square will rotate only.



In a dynamic block reference, if you change the value of a parameter associated with a polar stretch action through a grip or the Properties palette, it may affect the key point on the parameter associated with the polar stretch action. If the key point is affected, the geometry in the polar stretch action's selection set will move or rotate depending on how the block was defined.



If you only want objects in the block reference to rotate, don't include any objects in the stretch frame.

**See also:**

"Use Distance Multiplier and Angle Offset Action Overrides" on page 507

### To add a polar stretch action to a block definition

1. In the Block Editor, in the Block Authoring Palettes window, Actions tab, click the Stretch Action tool.
2. In the Block Editor drawing area, select a polar parameter to associate with the action. (You can associate only a polar parameter with a polar stretch action.)
3. Do one of the following to select a parameter point to associate with the action:
   - Use the pointing device.
   - Follow the prompts on the command line.

4 Do one of the following to specify the first corner of the stretch frame:

- Use the pointing device.
- Enter the *X* and *Y* coordinate values, separated by a comma.

5 Do one of the following to specify the opposite corner of the stretch frame:

- Use the pointing device.
- Enter the *X* and *Y* coordinate values, separated by a comma.

6 Select the objects to stretch or move.

7 Press ENTER.

8 Select the objects that you want to rotate.

(The objects you select in steps 6 and 8 make up the polar stretch action's selection set.)

9 Press ENTER.

10 (Optional) Follow the prompts on the command line to specify distance and offset.

11 Do one of the following to specify the location of the action:

- Use the pointing device.
- Enter the *X* and *Y* coordinate values, separated by a comma.

**NOTE** The action location in the block definition has no effect on the appearance or functionality of the block reference.

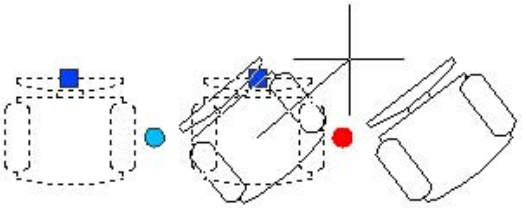
12 On the Block Editor toolbar, click Save Block Definition.

13 If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar

## Use a Rotate Action in a Dynamic Block

The rotate action is similar to the  command. In a dynamic block reference, a rotate action causes its associated objects to rotate when the associated parameter is edited through a grip or the Properties palette.



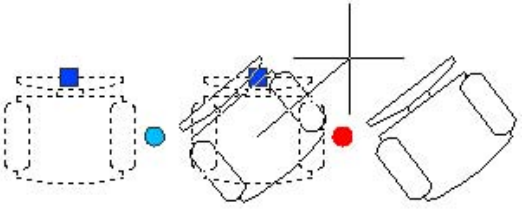
In a dynamic block definition, you can associate a rotate action only with a rotation parameter. The rotate action is associated with the entire parameter, not a key point on the parameter.

After associating a rotate action with a rotation parameter, you associate the action with a selection set of geometry.

#### Specify the Type of Base Point for a Rotate Action

The rotate action has a property called *Base Type*. With this property, you specify whether the base point for the rotation is the base point of the parameter or an independent base point that you specify in the block definition.

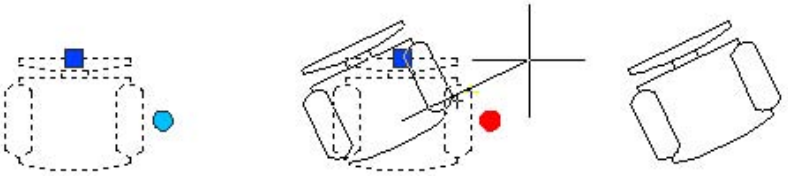
By default, Base Type is set to Dependent. This means that the block rotates around the associated rotation parameter's base point. In the following example, the chair block contains a rotation parameter and an associated rotate action. The rotate action's Base Type is Dependent. The base point of the parameter is at the center of the chair. Thus, the chair rotates about the center point.



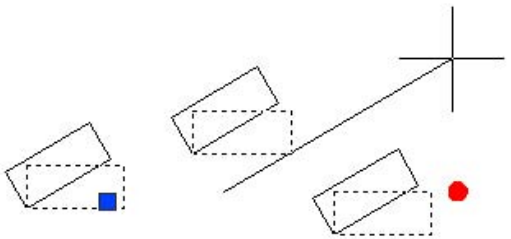
When you set Base Type to Independent, you can specify a base point for the rotate action other than the base point of the associated rotation parameter. This independent base point is shown in the Block Editor as an X marker. The location of the independent base point can be changed by dragging it or by editing the Base X and Base Y values in the Overrides section of the Properties palette.

In the following example, the chair block contains a rotation parameter and an associated rotate action. The rotate action's Base Type is Independent. The

independent base point is located at the lower left corner of the chair. Thus, the chair rotates about the lower left corner.



In the following example, each of the three rectangles in the dynamic block reference rotates about an independent base point located at the lower left corner of each rectangle. To achieve this, you assign one rotation parameter. You then add three rotate actions. Each rotate action is associated with the rotate parameter. Each rotate action is then associated with a different object and assigned a different independent base point.



You could achieve the same result by using dependent base points, each with a different base offset, for each rotation action. However, if you need to move the rectangles independently from one another (for example, with a polar or XY parameter and a move action) in the block reference, you should use independent base points for the rotate actions, or the objects won't rotate correctly.

**To add a rotate action to a block definition**

1. In the Block Editor, in the Block Authoring Palettes window, Actions tab, click the Rotate Action tool.
2. In the Block Editor drawing area, select a rotation parameter to associate with the action. (You can associate only a rotation parameter with a rotate action.)
3. Select the objects (the selection set) to associate with the action.

4. Press ENTER.
5. (Optional) Follow the prompts on the command line to specify the type of base point.
6. Do one of the following to specify the location of the action:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.

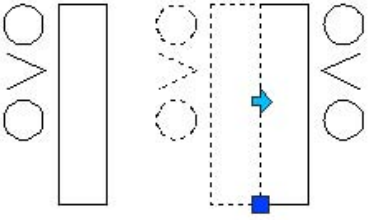
   **NOTE** The action location in the block definition has no effect on the appearance or functionality of the block reference.
7. On the Block Editor toolbar, click Save Block Definition.
8. If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar

## Use a Flip Action in a Dynamic Block

With a flip action, you can flip a dynamic block reference about a specified axis called a reflection line.

In a dynamic block reference, a flip action causes its associated selection set to flip about an axis called a *reflection line* when the associated parameter is edited through a grip or the Properties palette.



In a dynamic block definition, you can associate a flip action only with a flip parameter. The flip action is associated with the entire parameter, not a key point on the parameter. After associating a flip action with a flip parameter, you associate the action with a selection set of geometry. Only the selected objects will flip about the reflection line.

For example, in the following illustration, the polyline between the two circles is not included in the flip action's selection set. When the block reference is flipped, the polyline does not flip with the rest of the geometry.

**To add a flip action to a block definition**

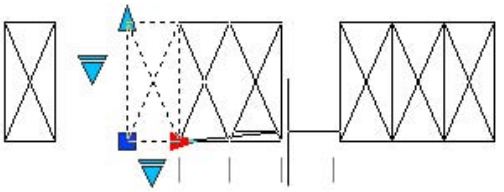
1. In the Block Editor, in the Block Authoring Palettes window, Actions tab, click the Flip Action tool.
2. In the Block Editor drawing area, select a parameter to associate with the action. (You can associate only a flip parameter with a flip action.)
3. Select the objects (the selection set) to associate with the action.
4. Press ENTER.
5. Do one of the following to specify the location of the action:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.

   **NOTE** The action location in the block definition has no effect on the appearance or functionality of the block reference.
6. On the Block Editor toolbar, click Save Block Definition.
7. If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar

## Use an Array Action in a Dynamic Block

In a dynamic block reference, an array action causes its associated objects to copy and array in a rectangular pattern when the associated parameter is edited through a grip or the Properties palette.



In a dynamic block definition, you can associate an array action with any of the following parameters:

- Linear
- Polar
- XY

After associating an array action with a parameter, you associate the action with a selection set of geometry.

### Specify Rows and Columns for an Array Action

When you associate an array action with a linear or polar parameter, you specify the column offset for the arrayed objects. The column offset determines the distance between the arrayed objects. When you edit the parameter in the block reference, the distance of the parameter (from base point to second point) is divided by the column offset to determine the number of columns (the number of objects).

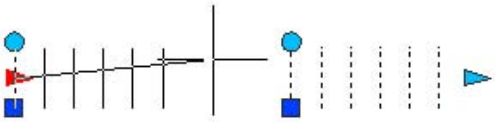
For example, you might associate an array action with a linear parameter. You specify that the array action has a column offset of 2. When you change the linear parameter in the dynamic block reference to a distance of 10, the number of columns for the block reference is 5.

If you associate an array action with an XY parameter, you can also specify the row offset.

### Including Parameters in an Array Action's Selection Set

When you include a parameter in an array action's selection set, it has no effect on the behavior of the block reference. The parameter is not copied with the other objects in the selection set. Additional grips are not displayed in the block reference.

In the following example, a parking lot block can be arrayed to contain any number of spaces. The vertical line can also be rotated. Notice that even after the block has been arrayed, it still contains only one rotation grip.



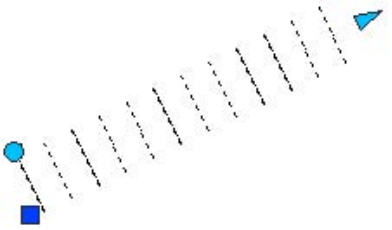
However, when you edit the grip for the parameter included in the array action's selection set, the parameter's associated action is triggered for all instances of the objects. The same behavior occurs when the parameter is not included in the array action's selection set.

### Using Rotate and Array Actions in the Same Dynamic Block

A dynamic block can contain an array action and a rotate action that have the same selection set. The order in which the block reference is arrayed and rotated affects the display of the block.

When you rotate the block first and then array it, all instances of the arrayed objects are individually rotated around their own base point.

When you array the block first and then rotate it, all instances of the arrayed objects are rotated around a single base point.



### To add an array action a block definition

1. In the Block Editor, in the Block Authoring Palettes window, Actions tab, click the Array Action tool.
2. In the Block Editor drawing area, select a parameter to associate with the action. (You can associate linear, polar, and XY parameters with an array action.)
3. Select the objects (the selection set) to associate with the action.
4. Press ENTER.

5 If you are associating the array action with an XY parameter, do one of the following:

- Enter the distance between rows.
- Specify a unit cell by entering two values separated by a comma or by picking two opposite points of a rectangle in the drawing area, and then go to step 7.

6 Enter a value for the distance between columns.

7 If you are assigning the array action to an XY parameter, enter the distance between rows.

8 Do one of the following to specify the location of the action:

- Use the pointing device.
- Enter the *X* and *Y* coordinate values, separated by a comma.

**NOTE** The action location in the block definition has no effect on the appearance or functionality of the block reference.

9 On the Block Editor toolbar, click Save Block Definition.

10 If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar

## Use a Lookup Action in a Dynamic Block

When you add a lookup action to a dynamic block definition and associate it with a lookup parameter, it creates a lookup table. You can use a lookup table to assign custom properties and values to a dynamic block.

For information about adding lookup tables to dynamic blocks, see "Use Lookup Tables to Assign Data to Dynamic Blocks" on page 521.

### To add a lookup action to a dynamic block definition

1 In the Block Editor, in the Block Authoring Palettes window, Actions tab, click the Lookup Action tool.

2 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.)

3 Do one of the following to specify the location of the action:

- Use the pointing device.
- Enter the *X* and *Y* coordinate values, separated by a comma.

**NOTE** The action location in the block definition has no effect on the appearance or functionality of the block reference.

4 In the Property Lookup Table dialog box, complete the table as necessary.

5 Click OK.

6 On the Block Editor toolbar, click Save Block Definition.

7 If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar

## Use Distance Multiplier and Angle Offset Action Overrides

Some actions have distance multiplier and angle offset override properties. These properties allow you to specify a factor by which a parameter value is increased or decreased when the parameter is edited in a dynamic block reference.

Action overrides are properties of actions. However, they have no effect on the block reference until the block reference is manipulated in a drawing. There are two types of action overrides: distance multiplier and angle offset.

- **Distance multiplier.** You 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.** You 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.

You can specify these action override properties by following the prompts on the command line 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.

The following table shows the available override properties for each type of action.

| Action Type | Available Override Properties |
|---|---|
| Move | Distance Multiplier, Angle Offset |
| Scale | None |
| Stretch | Distance Multiplier, Angle Offset |
| Polar Stretch | Distance Multiplier, Angle Offset |
| Rotate | None |
| Flip | None |
| Array | None |
| Lookup | None |

**To specify a distance multiplier override for an action**

1. In the Block Editor, select a move, stretch, polar stretch, or array action.
2. On the Properties palette, Overrides area, Distance Multiplier, enter a value.
3. Press ENTER.

**To specify an angle offset override for an action**

1. In the Block Editor, select a move, stretch, or polar stretch action.
2. On the Properties palette, Overrides area, Angle Offset, enter a value.
3. Press ENTER.

## Use Parameter Sets

Use the Parameter Sets tab on the Block Authoring palette to add commonly paired parameters and actions to your dynamic block definition.

You add a parameter set to your block the same way you add a parameter. The action included in the parameter set is automatically added to your block definition and associated with the added parameter. You must then associate a selection set (geometry) with each action.

When you first add a parameter set to your dynamic block definition, a yellow alert icon is displayed next to each action. This indicates that you need to

associate a selection set with each action. You can double click the yellow alert icon (or use the  command) and follow the prompts on the command line to associate the action with a selection set.

**NOTE** When you insert a lookup parameter set and double click the yellow alert icon, the Property Lookup Table dialog box is displayed. Lookup actions are associated with the data you add to this table, not a selection set.

The following table lists the parameter sets provided on the Parameter Sets tab of the Block Authoring palette.

| Parameter Set | Description |
|---|---|
| Point Move | Adds a point parameter with one grip and an associated move action to the dynamic block definition. |
| Linear Move | Adds a linear parameter with one grip and an associated move action to the dynamic block definition. |
| Linear Stretch | Adds a linear parameter with one grip and an associated stretch action to the dynamic block definition. |
| Linear Array | Adds a linear parameter with one grip and an associated array action to the dynamic block definition. |
| Linear Move Pair | Adds a linear parameter with two grips and a move action associated with each grip to the dynamic block definition. |
| Linear Stretch Pair | Adds a linear parameter with two grips and a stretch action associated with each grip to the dynamic block definition. |
| Polar Move | Adds a polar parameter with one grip and an associated move action to the dynamic block definition. |
| Polar Stretch | Adds a polar parameter with one grip and an associated stretch action to the dynamic block definition. |
| Polar Array | Adds a polar parameter with one grip and an associated array action to the dynamic block definition. |
| Polar Move Pair | Adds a polar parameter with two grips and a move action associated with each grip to the dynamic block definition. |
| Polar Stretch Pair | Adds a polar parameter with two grips and a stretch action associated with each grip to the dynamic block definition. |
| XY Move | Adds an XY parameter with one grip and an associated move action to the dynamic block definition. |
| XY Move Pair | Adds an XY parameter with two grips and move action associated with each grip to the dynamic block definition. |

| Parameter Set | Description |
|---|---|
| XY Move Box Set | Adds an XY parameter with four grips and a move action associated with each grip to the dynamic block definition. |
| XY Stretch Box Set | Adds an XY parameter with four grips and a stretch action associated with each grip to the dynamic block definition. |
| XY Array Box Set | Adds an XY parameter with four grips and an array action associated with each grip to the dynamic block definition. |
| Rotation | Adds a rotation parameter with one grip and an associated rotate action to the dynamic block definition. |
| Flip | Adds a flip parameter with one grip and an associated flip action to the dynamic block definition. |
| Visibility | Adds a visibility parameter with one grip. No action is required with a visibility parameter. |
| Lookup | Adds a lookup parameter with one grip and a lookup action to the dynamic block definition. |

**See also:**

"Use Block Authoring Palettes" on page 468

**To add parameter set to a dynamic block definition**

1. In the Block Editor, in the Block Authoring Palettes window, on the Parameter Sets tab, click a parameter set.
2. Follow the prompts on the command line.
3. On the Block Editor toolbar, click Save Block Definition.
4. (Optional) If you are finished using the Block Editor, click Close Block Editor.

## Specify Grips for Dynamic Blocks

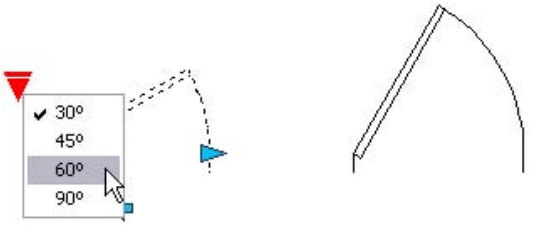
When you add a parameter to a dynamic block definition, custom grips associated with key points of the parameter are automatically added to the block. In a drawing, you can use these custom grips to manipulate the geometry of a dynamic block reference.

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.

If you specify that a parameter has 0 grips, you can still edit the dynamic block reference through the Properties palette (if the block is defined that way).

If a dynamic block definition contains visibility states or a lookup table, you can define the block so that the only grip that is displayed is a lookup grip. When you click this grip on the block reference, a drop-down list is displayed. When you select an item from the list, the display of the block reference may change.




Grips are automatically added at key points on the parameter. You can reposition a grip anywhere in the block space relative to its associated key point on the parameter. When you reposition a grip, it is still tied to the key point with which it is associated. No matter where the grip is shown in the block reference, it will still manipulate the key point with which it is associated. If you move or change the key point of the parameter, the grip's position relative to the key point is maintained. Because you use grips to manipulate dynamic block references in a drawing, you should make sure that each grip is placed in a logical location. If a grip is not in a logical place, the behavior may not be what is expected.

Both linear and polar parameters can display two, one, or no grips. When you define a linear or polar parameter to display one grip, it is displayed at the endpoint of the parameter. You should only assign actions to the endpoint of either of these parameters if you plan to display only one grip. Otherwise, you will not be able to manipulate the reference (trigger the actions) because a grip won't be displayed for the key point associated with the action.

If you reposition the grips for a dynamic block, you can use the  command to reset the grips to their default locations.

The type of parameter you add to the dynamic block definition determines the type of grips that are added to the block. These grips give visual clues as to how the block reference can be manipulated in a drawing. The following table details the types of grips used in dynamic blocks, what they look like, and the parameters with which they are associated.

| Grip Type | | How the Grip Can Be Manipulated in a Drawing | Associated Parameters |
|---|---|---|---|
| Standard | | Within a plane in any direction | Base, Point, Polar, and XY |
| Linear | | Back and forth in a defined direction or along an axis | Linear |
| Rotation | | Around an axis | Rotation |
| Flip | | Click to flip the dynamic block reference | Flip |
| Alignment | | Within a plane in any direction; when moved over an object, triggers the block reference to align with the object | Alignment |
| Lookup | | Click to display a list of items | Visibility, Lookup |

### Specify Insertion Cycling for Grips in Dynamic Blocks

Grips in dynamic blocks have a property called *Cycling*. When this property is set to Yes, the grip becomes an available insertion point for the dynamic block reference. You can use the  command to turn on and off cycling for grips in dynamic blocks as well as specify the cycling order for the grips. When you insert the dynamic block reference in a drawing, you can use the CTRL key to cycle through the available grips to select which grip will be the insertion point for the block.

#### To specify the number of custom grips displayed for a parameter in a dynamic block

1. In the Block Editor, select a parameter.
2. On the Properties palette, under Misc, click Number of Grips, and then select the number of grips you want to display for the parameter.
3. On the Block Editor toolbar, click Save Block Definition.
4. (Optional) If you are finished using the Block Editor, click Close Block Editor.

In the Block Editor, select a parameter. Right-click in the drawing area. Click Grip Display ➤ *any value*.

### To reposition a grip in a dynamic block definition

1. In the Block Editor, select a custom grip (not the standard grip associated with the key point of the parameter).
2. 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.)
3. On the Block Editor toolbar, click Save Block Definition.
4. (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. In the Block Editor, enter **bgripset** on the command line.
2. Select the grip for which you want to reset the grip position(s).
3. On the command line, enter **reposition**, and then press ENTER.
4. On the Block Editor toolbar, click Save Block Definition.
5. (Optional) If you are finished using the Block Editor, click Close Block Editor.

In the Block Editor, select a parameter. Right-click in the drawing area. Click Grip Display ➤ Reset Position.

### To turn insertion cycling on or off for a custom grip in a dynamic block reference

1. In the Block Editor, enter **bcycleorder** on the command line.
2. 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.)
3. Click OK.
4. On the Block Editor toolbar, click Save Block Definition.
5. (Optional) If you are finished using the Block Editor, click Close Block Editor.

In the Block Editor, select a dynamic grip. Right-click in the drawing area. Click Insertion Cycling.

**To modify the insertion cycling order for custom grips in a dynamic block reference**

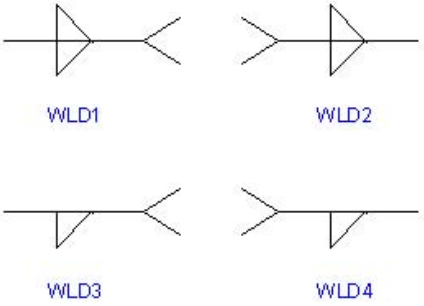
1. In the Block Editor, enter **bcycleorder** on the command line.
2. In the Insertion Cycling Order dialog box, select a grip from the list, and click Move Up or Move Down. (A check mark in the Cycling column indicates that cycling is turned on for the grip.)
3. Repeat step 2 until you are finished modifying the grip cycling order.
4. Click OK.
5. On the Block Editor toolbar, click Save Block Definition.
6. (Optional) If you are finished using the Block Editor, click Close Block Editor.

## Create Visibility States

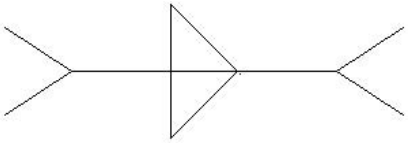
You can use visibility states to make geometry visible or invisible in a dynamic block. One block can have any number of visibility states.

Using visibility states is a powerful way to create a block that has many different graphical representations. You can easily change a block reference that has different visibility states without having to find a different one to insert in your drawing.

For example, you have the following four different weld symbols.

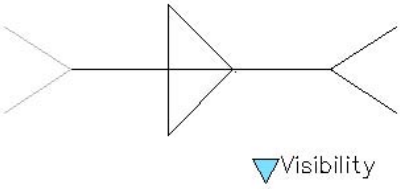



Using visibility states, you can combine these weld symbols into a single dynamic block. The following example shows the geometry for the four weld symbols combined in one dynamic block definition in the Block Editor.

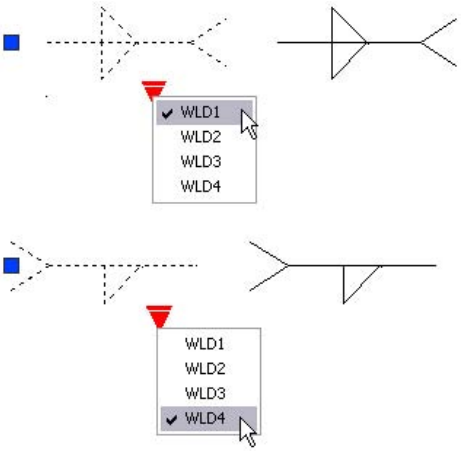


After you combine the geometry in the Block Editor, you add a visibility parameter. You add only one visibility parameter to a dynamic block definition. You do not associate any actions with a visibility parameter.

You can then create and name a different visibility state for each weld symbol (for example, WLD1, WLD2, WLD3, and WLD4). You can make particular geometry visible or invisible for each state. In the following example, the WLD1 visibility state is displayed in the Block Editor. The geometry that displays in a dimmed state is invisible for the WLD1 visibility state.




The visibility parameter includes a lookup grip. This grip is always displayed in a block reference that contains visibility states. When you click the grip in the block reference, a drop-down list of all the visibility states in the block reference is displayed. When you select one of the states from the list, the geometry that is visible for that state is displayed in the drawing.



The right-hand side of the Block Editor toolbar displays the name of the current visibility state. All blocks have at least one visibility state. You cannot delete the current state. This area of the tool bar also provides several tools for working with visibility states.

When you work with visibility states, you may or may not want to see the geometry that is invisible for a given state. You can use the Visibility Mode button () to display or not display geometry (in a dimmed state) that is invisible. When the BVMODE system variable is set to 1, geometry that is invisible for a given state displays in a dimmed mode.

**To enable visibility states (add a visibility parameter)**

1. In the Block Editor, in the Block Authoring Palettes window, Parameters tab, click the Visibility Parameter tool.
2. (Optional) Follow the prompts on the command line to specify the following for the parameter:
   - Name
   - Label
   - Description
   - Properties displayed for the block reference

   **NOTE** You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.
3. Do one of the following to specify a location for the parameter:
   - Use the pointing device.
   - Enter the *X* and *Y* coordinate values, separated by a comma.

   This is the default location for the visibility parameter grip.
4. On the Block Editor toolbar, click Save Block Definition.
5. (Optional) If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar

**To create a new visibility state based on the current state**

1. On the Block Editor toolbar, click Manage Visibility States.
2. In the Visibility States dialog box, click New.
3. In the New Visibility State dialog box, enter a name for the new visibility state.
4. Click Leave Visibility of Existing Objects Unchanged in New State.
5. Click OK.

The visibility of existing objects is unchanged in the new state.

Block Editor toolbar

**To create a new visibility state in which all objects are invisible**

1. On the Block Editor toolbar, click Manage Visibility States.
2. In the Visibility States dialog box, click New.
3. In the New Visibility State dialog box, enter a name for the new visibility state.
4. Click Hide All Existing Objects in New State.
5. Click OK.

Block Editor toolbar

**To create a new visibility state in which all objects are visible**

1. On the Block Editor toolbar, click Manage Visibility States.
2. In the Visibility States dialog box, click New.
3. In the New Visibility State dialog box, enter a name for the new visibility state.
4. Click Show All Existing Objects in New State.
5. Click OK.

Block Editor toolbar

**To make a visibility state current**

1. On the Block Editor toolbar, click the Visibility States control.
2. Select the visibility state that you want to set as the current state.

Block Editor toolbar

In the Visibility States dialog box, right-click a visibility state in the list. Click Set Current.

**To delete a visibility state**

1. On the Block Editor toolbar, click Manage Visibility States.
2. In the Visibility States dialog box, select a visibility state in the list.
3. Click Delete.
4. Click OK.

Block Editor toolbar

In the Visibility States dialog box, right-click a visibility state in the list. Click Delete State.

**To rename a visibility state**

1. In the Block Editor, click Manage Visibility States.
2. In the Visibility States dialog box, select a visibility state in the list.
3. Click Rename.
4. Enter a new name for the visibility state.
5. Click OK.

Block Editor toolbar

Right-click a visibility state in the list. Click Rename State.

**To make objects visible in the current visibility state**

1. On the Block Editor toolbar, click Make Visible.
2. In the drawing area, select the objects to make visible in the current visibility state.
3. Press ENTER.
4. On the command line, enter **current**.
5. Press ENTER.

Block Editor toolbar

In the Block Editor drawing area, select one or more objects. Right-click in the drawing area. Click Object Visibility ➤ Show for Current State.

**To make objects visible in all visibility states**

1. In the Block Editor drawing area, select the objects that you want to make visible in all visibility states.
2. Right-click in the drawing area.
3. Click Object Visibility ➤ Show for All States.

**To make objects invisible in the current visibility state**

1. On the Block Editor toolbar, click Make Invisible.
2. In the drawing area, select the objects that you want to make invisible in the current visibility state.
3. Press ENTER.
4. On the command line, enter **current**.
5. Press ENTER.

Block Editor toolbar

In the Block Editor drawing area, select one or more objects. Right-click in the drawing area. Click Object Visibility ➤ Hide for Current State.

**To make objects invisible in all visibility states**

1. In the drawing area, select the objects that you want to make invisible in all visibility states.
2. Right-click in the drawing area.
3. Click Object Visibility ➤ Hide for All States.

**To display or not display invisible geometry for visibility states**

1. On the Block Editor toolbar, click Visibility Mode.

Block Editor toolbar

**To change the order of the visibility states list for a dynamic block reference**

1. On the Block Editor toolbar, click Manage Visibility States.
2. In the Visibility States dialog box, select a visibility state in the list.
3. Click Move Up or Move Down to change the order.
4. Repeat steps 2 and 3 until the list of visibility states is in the correct order.
5. Click OK.

Block Editor toolbar

**To set the default visibility state for the dynamic block reference**

1. In the Block Editor, click Manage Visibility States.
2. In the Visibility States dialog box, select a visibility state in the list.

3 Click Move Up until the selected visibility state is at the top of the list.

4 Click OK.

Block Editor toolbar

# Use Lookup Tables to Assign Data to Dynamic Blocks

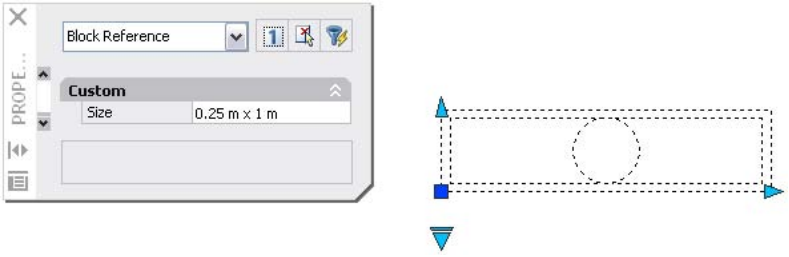
You can use a lookup table to define properties for and assign property values to a dynamic block.

## Overview of Using Lookup Tables to Assign Data to Dynamic Blocks

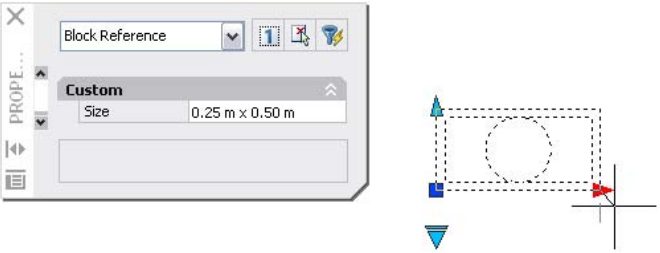
You can use a *lookup table* to define properties for and assign property values to a dynamic block. Using lookup tables is a powerful way to associate parameter values for the dynamic block reference with other data that you specify (for example, a model or part number). You can extract this data from block references in a drawing just as you would extract block attribute data.

When completed, the lookup table assigns property values to the dynamic block reference based on how it is manipulated in a drawing. Conversely, you can change how the block reference is displayed in the drawing by changing the value of a lookup property of the block reference through a lookup grip or the Properties palette.

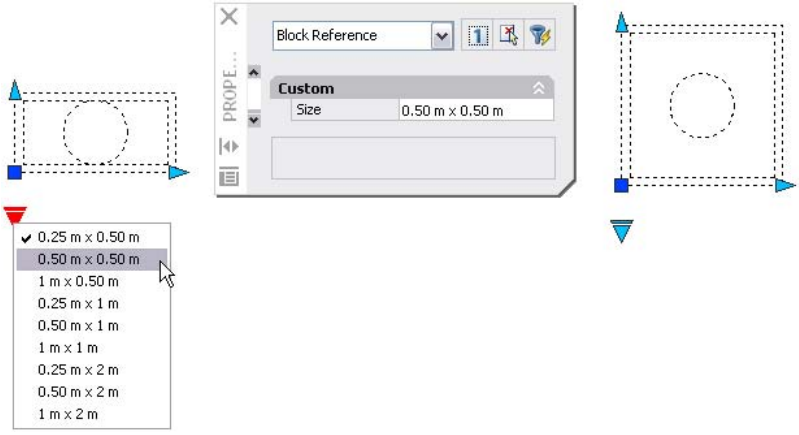
The following example shows a dynamic block reference for a lighting fixture. For this block, the Custom area of the Properties palette displays a size property. This property is defined in the lookup table for the block.




If you change the size of the lighting fixture in your drawing (using the custom grips), the size property in the Properties palette changes accordingly.



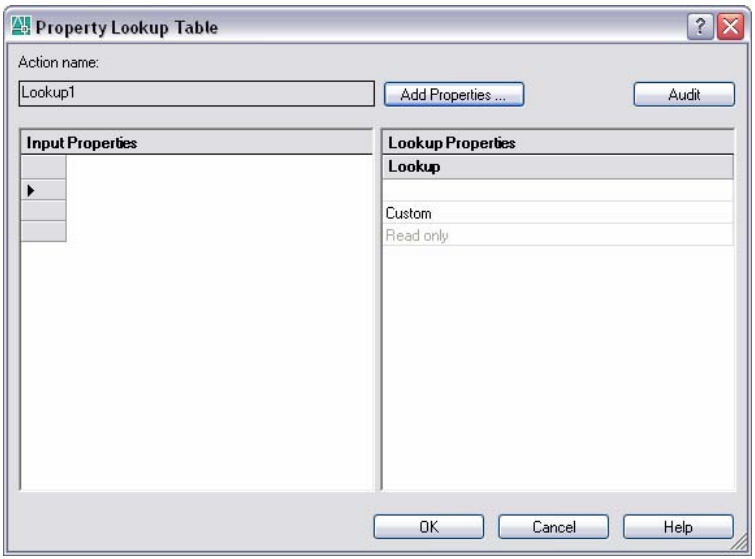

You can also use the lookup grip to change the display of the block reference. When you select a size from the list that is displayed when you click the grip, the block's geometry changes and the new size is displayed in the Properties palette under Custom.




### Create a Lookup Table

After you've drawn the geometry and added the appropriate parameters and actions required for the dynamic block's functionality, you can add a lookup table to the block definition.

A blank lookup table is created when you add a lookup parameter to a dynamic block definition and then add a lookup action and associate it with the lookup parameter. The lookup table is displayed in the lookup table dialog box.




### Add Properties and Values to a Lookup Table

The Property Lookup Table consists of input properties and lookup properties. Each input property and lookup property is represented by a column in the table.

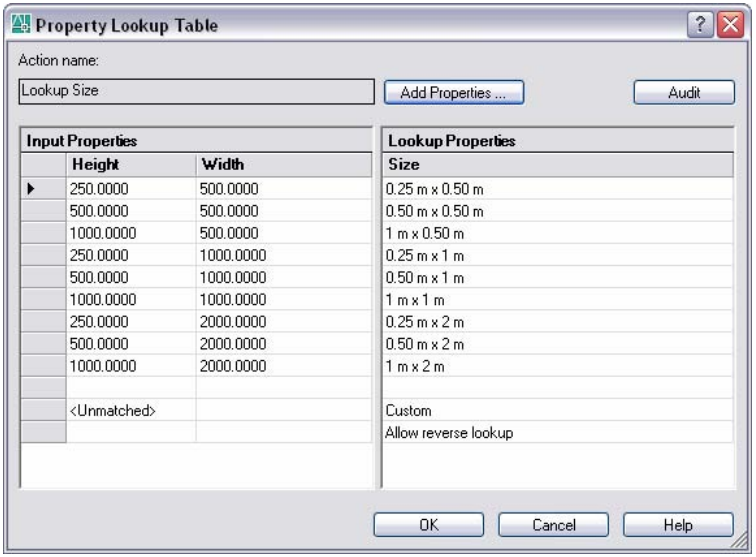
- **Input properties.** Parameters other than lookup parameters (for example, a linear parameter labeled "Width"). You can create a column under Input Properties for every parameter in the block definition except for lookup, alignment, and base point parameters.
- **Lookup properties.** Lookup parameters. You add one lookup parameter to the dynamic block definition for every lookup property column you want to add to the lookup table. The lookup parameter label is used as the property name.

After you add the properties (columns) to the table, you add values to the cells in each column. You click a cell and enter a value. Make sure to follow the guidelines in "Specify Values for Lookup Tables" on page 528.

If you defined a value set for an input property (parameter) in the table, a drop-down list of available values will display when you click a cell in that column.

The following lookup table is used for the lighting fixture block shown previously. The linear parameters that have been labeled "Height" and "Width" have been added as input properties. "Size" has been added as a lookup

property. The values under Height and Width are taken from the value sets defined for each of those parameters. The values under Size are entered manually.




When the parameter values in a dynamic block reference match a row of input property values, the corresponding lookup property values in that table row are assigned to the block reference. These lookup properties and values are displayed in the Properties palette under Custom.

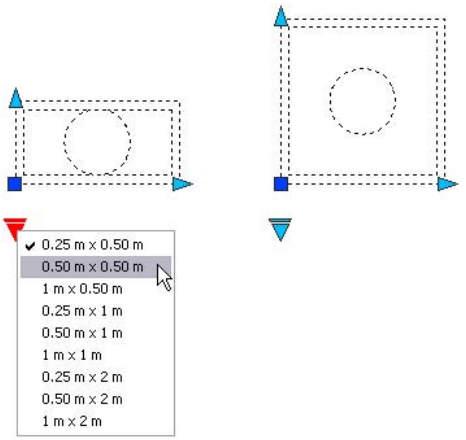
For example, when you edit the lighting fixture block reference to have a height of 250 centimeters and a width of 500 centimeters, the Size property in the Custom area of the Properties palette will display a value of 0.25 m x 0.50 m.

In the lookup table, at the bottom of the input properties is a row labeled <*Unmatched*>. When the parameter values in a block reference do not match a row of input property values defined in the block's lookup table, the lookup property value that corresponds to <Unmatched> is assigned to the block reference. The default value is *Custom*, but you can specify another value by changing it in the table.

### Enable Reverse Lookup

You can also specify that a lookup property allows for reverse lookup. This adds a lookup grip to the dynamic block reference. When you click this grip, a drop-down list of the lookup values for that lookup property (column in the table) is displayed. When you select a value from the list, the corresponding input property values are assigned to the block reference. Depending on how

the block was defined, this usually results in a change in the block reference's geometry.




To enable reverse lookup for a lookup property, each row in the lookup table must be unique. After you add properties and values to the table, you can check for errors and empty cells by clicking the audit button in the Property Lookup Table dialog box. You can also use the shortcut menu options to insert, delete, or reorder rows.

**NOTE** If you leave a cell empty in the lookup properties column and the cell has an associated input property in the same row, the lookup property will not allow for reverse lookup.

**See also:**

"**Extract Data from Block Attributes**" **on page 554**
"**Extract Block Attribute Data (Advanced)**" **on page 555**

### To create a lookup table for a dynamic block

1. In the Block Editor, add one or more lookup parameters to a dynamic block definition by following the steps in "To add a lookup parameter to a dynamic block definition" on page 486.
2. In the Block Editor, add a lookup action to a dynamic block definition by following the steps in "To add a lookup action to a dynamic block definition" on page 506.

   The Property Lookup Table dialog box is displayed.

3 Use any of the following procedures to set up a lookup table for the dynamic block:

- "To add input properties to a lookup table" on page 526
- "To add lookup properties to a lookup table" on page 526
- "To enable reverse lookup" on page 527

4 On the Block Editor toolbar, click Save Block Definition.

5 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar

### To add input properties to a lookup table

1 In the Block Editor, double-click a lookup action in a dynamic block definition.

2 In the Property Lookup Table dialog box, click Add Properties.

3 In the Add Parameter Properties dialog box, in the bottom-left corner of the dialog box, click Add Input Properties.

4 In the Parameter Properties list, select the parameter properties you want to add to the lookup table. Hold CTRL down to select more than one property.

5 Click OK.

### To add lookup 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 Lookup Properties.

4 In the Parameter Properties list, select the lookup parameter properties you want to add to the lookup table. Hold CTRL down to select more than one property.

5 Click OK.

**To add values 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 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.
3. Repeat Step 2 as required to complete the table.
4. Click OK.

**To enable reverse lookup**

1. In the Block Editor, double-click a lookup action in a dynamic block definition.
2. In the Property Lookup Table dialog box, at the bottom of a Lookup Property column, click Read Only, and then select Allow Reverse Lookup from the list.

   **NOTE** You can only select Allow Reverse Lookup if you have entered data in the column.
3. Click OK.

**To make the lookup property read-only**

1. In the Block Editor, double-click a lookup action in a dynamic block definition.
2. In the Property Lookup Table dialog box, at the bottom of a Lookup Property column, click Allow Reverse Lookup, and then select Read Only from the list.
3. Click OK.

**To check for errors in a lookup table**

1. In the Property Lookup Table dialog box, click Audit.

**To open an existing lookup table**

1 In the Block Editor, double-click a lookup action.

## Specify Values for Lookup Tables

The following rules apply when you specify values in lookup tables:

- Use a comma as the delimiter between values.
- You can specify any number of unique values separated by commas. For example: 5,6,7 5.5,6.25
- To specify a range, use brackets [ ] to specify that the range includes the values separated by a comma, or use parentheses ( ) to specify that the range does not include the values separated by a comma.
- For a continuous range, use a pair of values separated by a comma, enclosed in brackets or parentheses. For example: [3,10] specifies any value between 3 and 10, including 3 and 10 (3,10) specifies any value between 3 and 10, not including 3 and 10.
- For an open-ended range, use one value with a comma, enclosed in brackets or parentheses. For example: [,5] specifies less than or equal to 5; (5,) specifies greater than 5.
- Use no more than 256 characters in a table cell.
- You can use architectural and mechanical unit syntax (for example, 15'1/4").
- If you enter a value in an invalid format, the value will be reset to the last value when you move to another cell in the table.

Lookup tables support the following:

- All numeric parameter properties (for example, distance and angles for point, linear, polar, XY, and rotation parameters)
- Text string parameter properties (for example, flip and visibility parameter values)

No procedures for this topic.

# Specify Custom Properties for Dynamic Blocks

When you create a dynamic block, you can specify custom properties for the block. You can also specify whether or not these properties appear in the Properties palette when you select the block reference in a drawing.

## Overview of Specifying Custom Properties for Dynamic Blocks

In the Block Editor, you can specify properties for a parameter in a dynamic block definition. Some of these properties can be displayed as *custom properties* for the dynamic block reference when it is in a drawing. These properties are displayed under Custom in the Properties palette.

In the Block Editor, you can specify parameter labels. When you select the dynamic block reference in a drawing, these properties are shown under Custom in the Properties palette. It is good practice to specify unique parameter labels within the block.

Depending on the parameters used in the dynamic block definition, other parameter properties may be listed under Custom in the Properties palette when you select the dynamic block reference in a drawing. For example, a polar parameter has an angle property that displays in the Properties palette. Depending on how the dynamic block is defined, these properties might display values such as size, angle, and position for a selected block reference.

You can specify whether or not these custom properties are displayed for the block reference when it is selected in a drawing. These properties can also be extracted using the Attribute Extraction wizard.

You can also use the Properties palette to specify the geometric properties (such as color, linetype, and lineweight) of a parameter. These properties are listed in the Properties Palette under Geometry when you select a parameter in the Block Editor.

Other parameter properties, such as Value Set properties and Chain Actions, define how the block reference will function in a drawing.

In the Block Editor, you can also specify if the block can be exploded and if the block can be non-uniformly scaled.

**See also:**


- "Specify Value Sets for Dynamic Blocks" on page 532
- "Allow Chained Actions for Dynamic Blocks" on page 536
- "Specify Grips for Dynamic Blocks" on page 510
- "Extract Data from Block Attributes" on page 554

**To specify a parameter label**

1. In the Block Editor, click Tools menu ➤ Properties to display the Properties palette.
2. In the Block Editor, select a parameter.
3. In the Properties palette, under Property Labels, click <*parameter name*> label.
4. Enter a label for the parameter.
5. On the Block Editor toolbar, click Save Block Definition.
6. (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar

Right-click the selected object. Click Rename Label. Enter a new name for the parameter, and press ENTER.

**To specify the display of custom properties in a dynamic block reference**

1. In the Block Editor, click Tools menu ➤ Properties to display the Properties palette.
2. In the Block Editor, select a parameter.
3. In the Properties palette, under Misc, click Show Properties.
4. 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.
5. On the Block Editor toolbar, click Save Block Definition.
6. (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar

### To specify exploding dynamic block reference

1. Use one of the following procedures to open a block definition in the Block Editor:
   - "To open an existing block definition in the Block Editor" on page 464
   - "To open a block from the DesignCenter window in the Block Editor" on page 465
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. On the Block Editor toolbar, click Save Block Definition.
6. (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar

BEDIT

### To prevent non-uniform scaling in dynamic block reference

1. Use one of the following procedures to open a block definition in the Block Editor:
   - "To open an existing block definition in the Block Editor" on page 464
   - "To open a block from the DesignCenter window in the Block Editor" on page 465
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. On the Block Editor toolbar, click Save Block Definition.
6. (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar

BEDIT

## Specify Value Sets for Dynamic Blocks

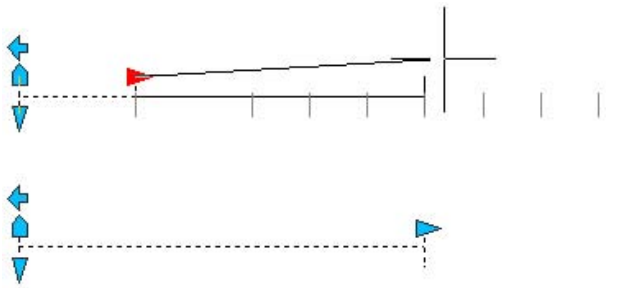
You can specify a defined *value set* for linear, polar, XY, and rotation parameters.

A value set is a range or list of values specified for a parameter. These values can be displayed for the block reference as a drop-down list next to the parameter label under Custom in the Properties palette. When you define a value set for a parameter, the parameter is limited to these values when the block reference is manipulated in a drawing. For example, if you define a linear parameter in a block that represents a window to have a value set of 20, 40, and 60, the window can only be stretched to 20, 40, or 60 units.

When you create a value list for a parameter, the value of the parameter as it exists in the definition is automatically added to the value set. This is the default value for the block reference when you insert it in a drawing.

In a block reference, if you change the parameter's value to a value other than one in the list, the parameter will adjust to the closest valid value. For example, you define a linear parameter to have a value set of 2, 4, and 6. When you try to change the value of that parameter in a block reference to 10, the resulting parameter value is 6 because it is the closest valid value.

When you specify a value set for a parameter in a dynamic block, tick marks are displayed when you grip-edit the block reference in a drawing. The tick marks indicate the locations for the parameter's valid values.



**NOTE** If you redefine the values in a value set after you've added the parameter properties to a lookup table, make sure to update the lookup table to match the new values in the value set.

**To specify a value set for a linear or polar parameter**

1. In the Block Editor, click Tools menu ➤ Properties to display the Properties palette.
2. In the Block Editor, select a linear or polar parameter.
3. In the Properties palette, under Value Set, click Dist Type.
4. On the drop-down list, select List.
5. In the Properties palette, under Value Set, click Dist Value List.
6. Click the [...] button.
7. In the Add Distance Value dialog box, under Distances to Add, enter one value or two or more values separated by commas.
8. Click Add.
9. Click OK.
10. On the Block Editor toolbar, click Save Block Definition.
11. (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar

**To specify a value set for an XY parameter**

1. In the Block Editor, click Tools menu ➤ Properties to display the Properties palette.
2. In the Block Editor, select an XY parameter.
3. In the Properties palette, under Value Set, click Hor Type or Ver Type.
4. On the drop-down list, select List.
5. In the Properties palette, under Value Set, click Hor Value List or Ver Value List.
6. Click the [...] button.
7. In the Add Distance Value dialog box, under Distances to Add, enter one value or two or more values separated by commas.
8. Click Add.
9. Click OK.

10 On the Block Editor toolbar, click Save Block Definition.

11 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar

**To specify a value set for a rotation parameter**

1 In the Block Editor, click Tools menu ➤ Properties to display the Properties palette.

2 In the Block Editor, select a rotation parameter.

3 In the Properties palette, under Value Set, click Ang Type.

4 On the drop-down list, select List.

5 In the Properties palette, under Value Set, click Ang Value List.

6 Click the [...] button.

7 In the Add Angle Value dialog box, under Angle to Add, enter one value or two or more values separated by commas.

8 Click Add.

9 Click OK.

10 On the Block Editor toolbar, click Save Block Definition.

11 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar

**To delete values from a parameter's value set**

1 In the Block Editor, click Tools menu ➤ Properties to display the Properties palette.

2 In the Block Editor, select a linear, polar, XY, or rotation parameter.

3 In the Properties palette, under Value Set, click one of the following:

- Dist Value List

- Ang Value List
- Hor Value List
- Ver Value List

4. Click the [...] button.
5. In the Add Distance Value or Add Angle Value dialog box, select a value from the list to delete.
6. Click Delete.
7. Click OK.
8. On the Block Editor toolbar, click Save Block Definition.
9. (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar

**To specify an incremental value set for a linear, polar, XY, or rotation parameter**

1. In the Block Editor, click Tools menu ➤ Properties to display the Properties palette.
2. In the Block Editor, select a linear, polar, XY, or rotation parameter.
3. In the Properties palette, under Value Set, click Dist Type, Ang Type, Hor Type, or Ver Type.
4. On the drop-down list, select Increment.
5. 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.
6. Click Dist Minimum, Ang Minimum, Hor Minimum, or Ver Minimum, and then enter a minimum value for the parameter.
7. Click Dist Maximum, Ang Maximum, Hor Maximum, or Ver Maximum, and then enter a maximum value for the parameter.
8. On the Block Editor toolbar, click Save Block Definition.
9. (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar

**To specify minimum and maximum values for a linear, polar, XY, or rotation parameter**

1. In the Block Editor, click Tools menu ➤ Properties to display the Properties palette.
2. In the Block Editor, select a linear, polar, XY, or rotation parameter.
3. 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.
4. Click Dist Maximum, Ang Maximum, Hor Maximum, or Ver Maximum, and then enter a maximum value for the parameter.
5. On the Block Editor toolbar, click Save Block Definition.
6. (Optional) If you are finished using the Block Editor, click Close Block Editor.

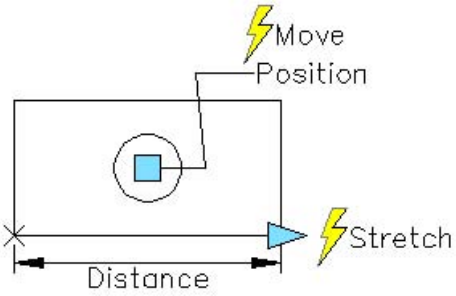
Standard toolbar

## Allow Chained Actions for Dynamic Blocks

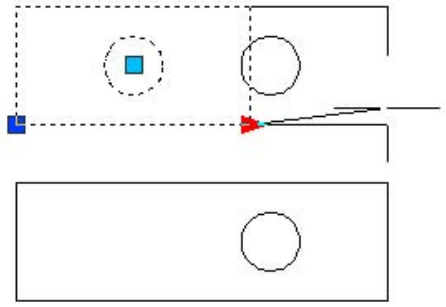
Point, linear, polar, XY, and rotation parameters have a property called *Chain Actions*. This property affects the parameter behavior if the parameter is part of an action's selection set.

For example, you might include a point parameter in the selection set of a stretch action that is associated with a linear parameter. When the linear parameter is edited in a block reference, its associated stretch action triggers a change in its selection set. Because the point parameter is included in the selection set, the point parameter is edited by the change in the linear parameter.

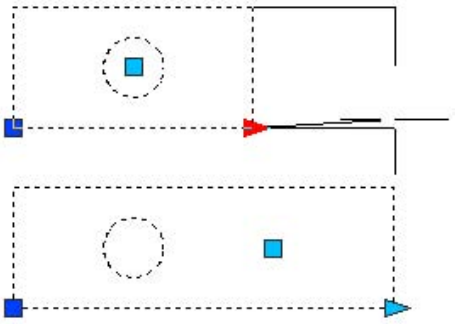
The following example shows a block definition in the Block Editor. The point parameter (labeled Position) is included in the stretch action's selection set.




If the Chain Actions property for the point parameter is set to Yes, a change in the linear parameter will trigger the move action associated with the point parameter, just as if you edited the point parameter in the block reference through a grip or custom property.



If the Chain Actions property is set to No, the point parameter's associated move action is not triggered by the changes to the linear parameter. Thus, the circle doesn't move.



**To specify the Chain Actions property for a point, linear, polar, XY, or rotation parameter**

1. In the Block Editor, click Tools menu ➤ Properties to display the Properties palette.
2. In the Block Editor, select a point, linear, polar, XY, or rotation parameter.
3. In the Properties palette, under Misc, click Chain Actions.
4. On the drop-down list, select Yes or No.
5. On the Block Editor toolbar, click Save Block Definition.
6. (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar

## Modify Dynamic Block Definitions

After you've defined a dynamic block, you can modify it in the Block Editor. You can delete, add, and modify the following elements in the Block Editor:

- Parameters (see "Use Parameters in Dynamic Blocks" on page 476)
- Actions (see "Use Parameters in Dynamic Blocks" on page 476 "Use Actions with Dynamic Blocks" on page 488)
- Geometry
- Visibility states (see "Create Visibility States" on page 514)
- Lookup tables (see "Use Lookup Tables to Assign Data to Dynamic Blocks" on page 521)
- Grips (see "Specify Grips for Dynamic Blocks" on page 510)
- Properties (see "Overview of Specifying Custom Properties for Dynamic Blocks" on page 529)

You can also use the  command to change an action's selection set. If you delete a parameter associated with an action and need to reassign that action to another parameter, use the  command.

After you modify a dynamic block definition in the Block Editor, you must save your changes (see "Save a Block in the Block Editor" on page 541).

**WARNING** If you redefine a dynamic block in AutoCAD 2005 or earlier, the block will lose its dynamic behavior.

**To open an existing dynamic block definition in the Block Editor**

1. Open the drawing file that contains the dynamic block definition, or open the drawing file that has been saved as a block.
2. Follow the steps in "To open an existing block definition in the Block Editor" on page 464.

Standard toolbar

Right-click a selected block. Click Block Editor.

**To delete an element from a dynamic block definition**

1. In the Block Editor, select the element you want to delete from the dynamic block definition.
2. Press the DELETE key.

**To modify an action's selection set in a dynamic block definition**

1. In the Block Editor, select an action.
2. On the command line, enter **bactionset**, and then press ENTER.
3. Follow the prompts on the command line.
4. On the Block Editor toolbar, click Save Block Definition.
5. (Optional) If you are finished using the Block Editor, click Close Block Editor.

**To redefine a stretch or polar stretch action's stretch frame set in a dynamic block definition**

1. In the Block Editor, double-click a stretch or polar stretch action.
2. Follow the prompts on the command line to define a new stretch frame and add objects to or remove objects from the action's selection set.
3. On the Block Editor toolbar, click Save Block Definition.

4 (Optional) If you are finished using the Block Editor, click Close Block Editor.

## Correct Errors in Dynamic Block Definitions

A yellow alert icon is displayed in the Block Editor if a dynamic block definition contains errors or is incomplete. You need to correct the errors (or complete the block) so the block reference will function properly in a drawing.

The yellow alert icon indicates that the block definition is not defined correctly or is incomplete. For example, a parameter that is not associated with an action will display an alert icon. An action that is not associated with a parameter or a selection set, will also display an alert icon.

You can correct these errors by double-clicking the yellow alert icon and following the prompts on the command line.

After you create a dynamic block definition in the Block Editor, you should save it and then test the functionality of the block reference in a drawing. If the block reference isn't functioning the way you intended, open the definition in the Block Editor and double-check the types of parameters and actions you used as well as their dependencies and properties.

**To correct an error indicated by a yellow alert icon in a dynamic block definition**

1 In the Block Editor, double-click a yellow alert icon.

2 Follow the prompts on the command line to correct the errors in the dynamic block definition.

3 On the Block Editor toolbar, click Save Block Definition.

4 (Optional) If you are finished using the Block Editor, click Close Block Editor.

**To view dependencies within a dynamic block definition**

1 In the Block Editor, select a parameter or an action.

If you selected a parameter, its associated action is highlighted.

If you selected an action, its associated parameter and selection set of geometry is highlighted.

2 When you're finished viewing the dependencies, press ESC.

3 (Optional) If you are finished using the Block Editor, click Close Block Editor.

# Save a Block in the Block Editor

When you are finished adding elements to your dynamic block definition in the Block Editor, save the block definition.

In the Block Editor, you can save your block definition by clicking the Save Block Definition button on the Block Editor toolbar, or by entering **bsave** on the command line. You should then save your drawing to make sure that the block definition is saved in the drawing.

When you save a block definition in the Block Editor, the current values of the geometry and parameters in the block are set as the default values for the block reference. When you create a dynamic block that uses visibility states, the default visibility state for the block reference is the visibility state at the top of the list in the Manage Visibility States dialog box.

Once you've saved the block definition, you can close the Block Editor and try your block in a drawing.

---

**NOTE** If you click File ➤ Save while you are in the Block Editor, you will save the drawing but not the block definition. You must specifically save the block definition while you are in the Block Editor.

---

**To save a block definition in the Block Editor**

1. On the Block Editor toolbar, click Save Block Definition.
2. Click File ➤ Save to save the block definition in the drawing.
3. (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar

**To save a copy of the current block definition in the Block Editor under a new name**

1. On the Block Editor toolbar, click Save Block Definition As.
2. In the Save Block As dialog box, enter a name for the new block definition.
3. Click OK.
4. Click File ➤ Save to save the block definition in the drawing.

5 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar

# Control the Color and Linetype Properties in Blocks

The objects in an inserted block can retain their original properties, can inherit properties from the layer on which they are inserted, or can inherit the properties set as current in the drawing.

Generally when you insert a block, the color, linetype, and lineweight of objects in the block retain their original settings regardless of the current settings in the drawing. However, you can create blocks with objects that inherit the current color, linetype, and lineweight settings. These objects have floating properties.

You have three choices for how the color, linetype, and lineweight properties of objects are treated when a block reference is inserted.

- Objects in the block do not inherit color, linetype, and lineweight properties from the current settings. The properties of objects in the block do not change regardless of the current settings.

  For this choice, it is recommended that you set the color, linetype, and lineweight properties individually for each object in the block definition: do not use BYBLOCK or BYLAYER color, linetype, and lineweight settings when creating these objects.

- Objects in the block inherit color, linetype, and lineweight properties from the color, linetype, and lineweight assigned to the current layer only.

  For this choice, before you create objects to be included in the block definition, set the current layer to 0, and set the current color, linetype, and lineweight to BYLAYER.

- Objects inherit color, linetype, and lineweight properties from the current color, linetype, and lineweight that you have set explicitly, that is, that you have set to override the color, linetype, or lineweight assigned to the current layer. If you have not explicitly set them, then these properties are inherited from the color, linetype, and lineweight assigned to the current layer.

For this choice, before you create objects to be included in the block definition, set the current color or linetype to BYBLOCK.

| If you want objects in a block to | Create objects on these layers | Create objects with these properties |
|---|---|---|
| Retain original properties | Any but 0 (zero) | Any but BYBLOCK or BYLAYER |
| Inherit properties from the current layer | 0 (zero) | BYLAYER |
| Inherit individual properties first, then layer properties | Any | BYBLOCK |

Floating properties also apply to nested blocks when the nested block references and the objects they contain use the settings required for floating properties.

**To set the color for all new objects**

1. On the Properties toolbar, click the Color control.
2. Either click a color to draw all new objects in that color, or click Select Color to display the Select Color dialog box and do one of the following:
   - On the Index tab, click a color or enter the ACI color number (1-255) or name in the Color box, and then click OK.
   - On the True Color tab, select the HSL color model in the Color Model option and specify a color by entering a color value in the Color box or by specifying values in the Hue, Saturation, and Luminance boxes, and then click OK.
   - On the Color Books tab, select a color book from the Color Book box, select a color by navigating the color book (using the up and down arrows) and clicking on a color chip, and then click OK.
   - Click BYLAYER to draw new objects in the color assigned to the current layer.
   - Click BYBLOCK to draw new objects in the current color until they are grouped into a block. When the block is inserted into the drawing, the objects in the block acquire the current color setting.
3. Click OK.

   The Color control displays the current color.

**To set the linetype for all new objects**

1. On the Format menu, click Linetype.
2. If you need to load additional linetypes, click Load, select one or more linetypes, and click OK.

   You can hold down CTRL to select several linetypes or SHIFT to select a range of linetypes.
3. In the Linetype Manager, do one of the following:

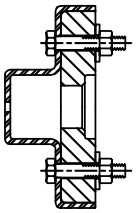
   - Select a linetype and select Current to draw all new objects with that linetype.
   - Select BYLAYER to draw new objects in the linetype assigned to the current layer.
   - Select BYBLOCK to draw new objects in the current linetype until they are grouped into a block. When the block is inserted in a drawing, the objects in the block acquire the current linetype setting.
4. Click OK.

   The Linetype control displays the current linetype. If the linetype you want to use is already loaded, you can click the Linetype control and click the linetype to make it current.

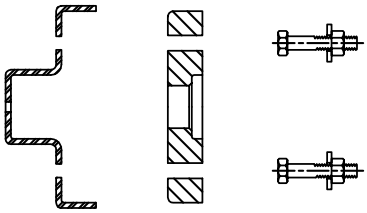
# Nest Blocks

Block references that contain other blocks are known as nested blocks. Using blocks within blocks can simplify the organization of a complex block definition.

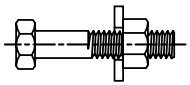
With nested blocks, you can build a single block out of several components. For example, you can insert as a block a drawing of a mechanical assembly that contains a housing, a bracket, and fasteners in which each fastener is a block composed of a bolt, washer, and nut. The only restriction on nested blocks is that you cannot insert blocks that reference themselves.



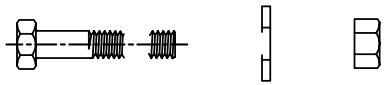
assembly block



blocks that are components of the assembly block



fastener block



blocks that are components of the fastener block

Procedures to come.

# Insert Blocks

When you insert a block, you create a block reference and specify its location, scale, and rotation.

When you insert a block, you create a block reference. You determine its location, scale factor, and rotation angle. You can specify the scale of a block reference using different *X, Y,* and *Z* values. Inserting a block creates an object called a block reference because it references a block definition stored in your current drawing. An orange lightning bolt icon displayed in the bottom-right corner of the block preview indicates that the block is dynamic.

default values

X scale = .5
Y scale = 1

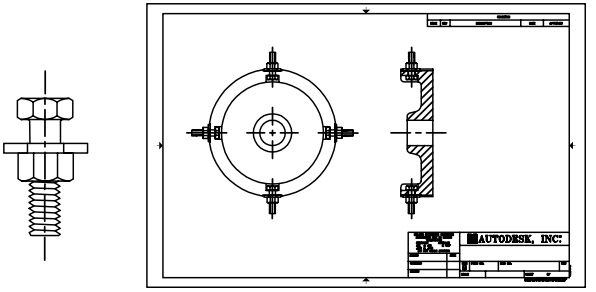
X scale = 1
Y scale = .5

rotation angle
= 45

If you insert a block that uses different drawing units than the units specified for the drawing, the block is automatically scaled by a factor equivalent to the ratio between the two units.

### Insert a Drawing File as a Block

When you insert an entire drawing file into another drawing, the drawing information is copied into the block table of the current drawing as a block definition. Subsequent insertions reference the block definition with different position, scale, and rotation settings, as shown in the following illustration.

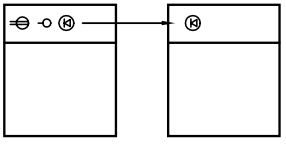



fastener block instances of fastener

Xrefs contained in a drawing you insert may not be displayed properly unless the xref was previously inserted or attached to the destination drawing.

### Insert Blocks from Block Libraries

You can insert one or more block definitions from an existing drawing file into your current drawing file. Choose this method when retrieving blocks from block library drawings. A block library drawing contains block definitions of symbols with similar functions. These block definitions are stored together in a single drawing file for easy accessibility and management.



block definition inserted from block library drawing

### Insert Blocks at Intervals

You can insert blocks at intervals along a selected geometric object.

■ Use to insert a block at measured intervals.

■ Use  to insert a block at proportional (evenly spaced) intervals.

### Insert Blocks with DesignCenter

Use DesignCenter to insert blocks from the current drawing or from another drawing. Drag and drop the block names for quick placement. Double-click the block names to specify the precise location, rotation, and scale of the blocks.

You cannot add blocks to a drawing while another command is active, and you can only insert or attach one block at a time.

**See also:**


"Create Drawing Files for Use as Blocks" on page 456
"Overview of Blocks" on page 452
"Create Block Libraries" on page 455
"Work with Dynamic Blocks in Drawings" on page 548
"Add Content with DesignCenter" on page 67
"Create and Use Tools from Objects and Images" on page 22


### To insert a block defined in the current drawing

1. On the Insert menu, click Block.
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.

Insert toolbar

### 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. On the Tools menu, click DesignCenter.

   The DesignCenter window is displayed.

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:

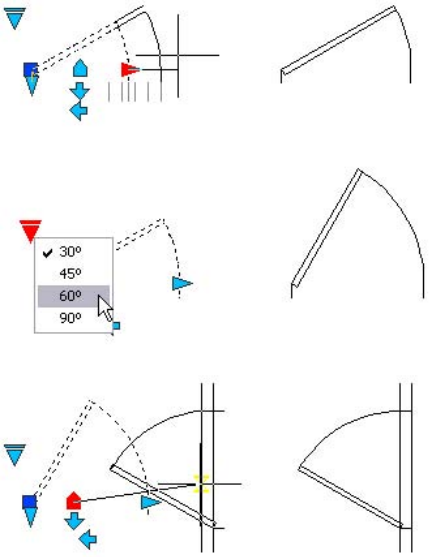
   - Drag the drawing file or block into your current drawing. Use this option when you want to insert blocks quickly and move or rotate the blocks to their precise locations later.
   - Double-click the drawing file or block that you want to insert into your current drawing. Use this option when you want 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.

Standard toolbar

# Work with Dynamic Blocks in Drawings

A *dynamic* block has flexibility and intelligence. A dynamic block reference can easily be changed in a drawing while you work. You can manipulate the geometry through custom grips or custom properties. This allows you to adjust the block reference in-place as necessary rather than searching for another block to insert or redefining the existing one.

For example, if you insert a door block reference in a drawing, you might need to change the size of the door while you're editing the drawing. If the block is dynamic and defined to have an adjustable size, you can change the size of the door simply by dragging the custom grip or by specifying a different size in Properties palette. You might also need to change the open angle of the door. The door block might also contain an alignment grip, which allows you to align the door block reference easily to other geometry in the drawing.




A dynamic block can have custom grips and custom properties. Depending on how the block was defined, you may be able to manipulate the block through these custom grips and custom properties. By default, custom grips for a dynamic block are a different color than standard grips. You can change the display color for custom grips with the  system variable. The following table shows the different types of custom grips that can be included in a dynamic block.

| Grip Type | How the Grip Can Be Manipulated in a Drawing |
|---|---|
| Standard | Within a plane in any direction |
| Linear | Back and forth in a defined direction or along an axis |
| Rotation | Around an axis |
| Flip | Clicked to flip the dynamic block reference |
| Alignment | Within a plane in any direction; when moved over an object, triggers the block reference to align with the object |

| Grip Type | How the Grip Can Be Manipulated in a Drawing |
| --- | --- |
| Lookup | Clicked to display a list of items |

After you've manipulated a dynamic block in a drawing, you can reset it. When you reset a block reference, the block changes back to the default specified in the block definition. If you non-uniformly scale or explode a dynamic block reference, it loses its dynamic properties. You can reset the block to its default values, which will make it dynamic again.

Some dynamic blocks are defined so that geometry within the block can only be edited to certain sizes specified in the block definition. When you use a grip to edit the block reference, tick marks are displayed at the locations of valid values for the block reference. If you change a block property value to a value other than one specified in the definition, the parameter will adjust to the closest valid value. For example, a block is defined to have a length of 2, 4, and 6. When you try to change that distance value to 10, the resulting value is 6 because it is the closest valid value.

**To manipulate a dynamic block using custom grips**

1. In a drawing, select a dynamic block reference.
2. Use the grips to stretch or change the block.

**To manipulate a dynamic block using custom properties**

1. In a drawing, select a dynamic block reference.
2. In the Properties palette, under Custom, change the required values.

**To reset block references in a drawing**

1. In a drawing, select a dynamic block reference.
2. Right-click in the drawing area. Click Reset Block.

**NOTE** To reset more than one block reference at a time, use the  command.

**To change the display color for custom grips**

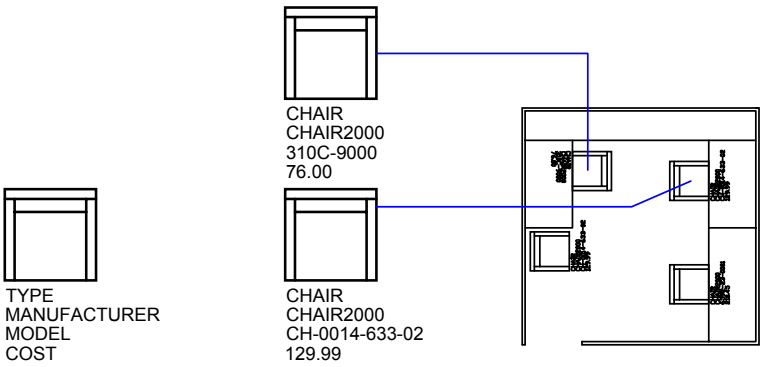
1. On the command line, enter **gripdyncolor**.
2. Enter an integer from 1 to 255 (ACI color).
3. Press ENTER.

# 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 tags for type, manufacturer, model, and cost.




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.

Whenever you insert a block that has a variable attribute, you are prompted to enter data to be stored with the block. Blocks may also use constant attributes, attributes whose values do not change. Constant attributes do not prompt you for a value when you insert the block.

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.

**See also:**

"Modify Block Attributes"

# Define Block Attributes

To create an attribute, you first create an attribute definition, which describes the characteristics of the attribute. 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, and any optional modes (Invisible, Constant, Verify, and Preset).

After creating the attribute definition, you select it as one of the objects when you define the block. Then, whenever you insert the block, you are prompted with the text you specified for the attribute. For each new block insertion, you can specify a different value for the attribute.

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.

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.

### Correct Mistakes in Block Attribute Definitions

If you make a mistake, you can use the Properties palette or  to edit an attribute definition before it is associated with a block. You can change the tag, prompt, and default value.

### Attach Attributes to Blocks

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 in the selection set any attributes you want to attach to the block.

To attach several attributes to the same block, first define the attributes and then include them in the block definition. The order in which you select the attributes determines the order in which you are prompted for attribute information when you insert the block.

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're working in the Block Editor, you can also use the Attribute Order dialog box to change the order in which you are prompted for attribute information when you insert the block reference. You can only do this when you have a block definition open in the Block Editor.

#### 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. On the Draw menu, click Block ➤ Define Attributes.
2. In the Attribute Definition dialog box, set the attribute modes and enter tag information, location, and text options.
3. Click OK.

   After creating the attribute definition, you can select it as an object while creating a block definition. If the attribute definition is incorporated into a block, whenever you insert the block, you are prompted with the text string you specified for the attribute. Each subsequent instance of the block can have a different value specified for the attribute.

#### To edit an attribute definition before it is associated with a block

1. On the Modify menu, click 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 order of attribute definitions

1. In the Block Editor, select a block attribute.
2. Right-click in the Block Editor drawing area.

3. Click Attribute Order.
4. In the Attribute Order dialog box, select an attribute definition.
5. Click Move Up or Move Down to change the order of the attribute definitions.
6. Repeat steps 2 and 3 until the attribute definition list is in the desired order.
7. Click OK.

➤

# Extract Data from Block Attributes

If you have attached attributes to blocks, you can then query one or more drawings for this block attribute information and save it to a table or to an external file.

Extracting attribute information is an easy way to produce a schedule or bill of materials directly from your drawing data. For example, a facilities drawing might contain blocks representing office equipment. If each block has attributes identifying the model and manufacturer of the equipment, you can generate a report that estimates the cost of the equipment.

The Attribute Extraction wizard guides you through selecting drawings, block instances, and attributes. The wizard can also create a file with a *.blk* file extension that contains all the settings for later reuse.

### Output to a Table

If you extract attribute data to a table, the table is inserted in the current drawing and current space (model space or paper space) and on the current layer.

When you update the table, the attribute information is extracted again and the data rows in the table are replaced. If you have included a title row, one or more header rows, or a summary row in the table, they are not replaced during update. You can insert mathematical expressions in the cells of the summary row that do calculations using values in the table cells.

---

**NOTE** For access to shortcut menus in the drawing area that are needed for editing and updating tables, the Shortcut Menus in Drawing Area must be checked in the Options dialog box, User Preferences tab.

---

#### Output to a File

If you save the data to an external file, the comma-separated (CSV), tab-separated (TXT), Microsoft Excel (XLS), and Microsoft Access (MDB) file formats are available.

When the characters period (.), comma (,), or pound sign (#) are written to an Excel or Access file, they are replaced with their Unicode representation.

#### To extract block attributes to a table or a file

- Click Tools menu ➤ Attribute Extraction.

  The Attribute Extraction Wizard is displayed. The wizard provides step-by-step instructions for extracting information from block attributes in the current drawing or other drawings. The information is used to create a table in the current drawing or is saved to an external file.

  The "Attribute Extraction Wizard" topic describes the options in the wizard.

Modify II toolbar

#### To update an attribute extraction table

- When the "Attribute Extraction Table Needs Updating" notification is displayed in the status tray, click Refresh Table Data.

#### To turn off update notification for attribute extraction tables

1. To turn off notification for all notification-enabled tables in the drawing, right-click the notification icon in the status tray. Clear the Enable Notifications for Out-of-Synch Table Data option.
2. To turn off notification for a selected table, right-click. Click the Suppress Notifications for This Table option.

The notification option is on by default on the Table Style page when you use the Attribute Extraction wizard.

# Extract Block Attribute Data (Advanced)

You can extract attribute information from a drawing and create a separate text file for use with database software. This feature is useful for creating parts

lists with information already entered in the drawing database. Extracting attribute information does not affect the drawing.

To create a parts list

- Create and edit an attribute definition
- Enter values for the attributes as you insert the blocks
- Create a template file and then extract attribute information to a text file

To extract attribute information, you first create an attribute template file using any text processor, then generate the attribute extraction file using AutoCAD, and, finally, open the attribute extraction file in a database application. If you plan to extract the attribute information to a DXF (drawing interchange format) file, it is not necessary to first create an attribute template file.

---

**NOTE** Make sure that the attribute extraction file does not have the same name as the attribute template file.

---

#### Create an Attribute Extraction Template File

Before you extract attribute information, you must create an ASCII template file to define the structure of the file that will contain the extracted attribute information. The template file contains information about the tag name, data type, field length, and number of decimal places associated with the information you want to extract.

Each field in the template file extracts information from the block references in the drawing. Each line in the template file specifies one field to be written to the attribute extraction file, including the name of the field, its character width, and its numerical precision. Each record in the attribute extraction file includes all the specified fields in the order given by the template file.

The following template file includes the 15 possible fields. *N* means numeric, *C* means character, *www* means a 3 digit number for the total width of the field, and *ddd* means a 3 digit number representing how many numeric decimal places are to be displayed to the right of the decimal point.

BL:NAME *Cwww*000 *(Block name)*
BL:LEVEL *Nwww*000 *(Block nesting level)*
BL:X *Nwwwddd (X coordinate of block insertion point)*
BL:Y *Nwwwddd (Y coordinate of block insertion point)*
BL:Z *Nwwwddd (Z coordinate of block insertion point)*
BL:NUMBER *Nwww*000 *(Block counter; the same for MINSERT)*
BL:HANDLE *Cwww*000 *(Block handle; the same for MINSERT)*
BL:LAYER *Cwww*000 *(Block insertion layer name)*

BL:ORIENT *Nwwwddd (Block rotation angle)*
BL:XSCALE *Nwwwddd (X scale factor)*
BL:YSCALE *Nwwwddd (Y scale factor)*
BL:ZSCALE *Nwwwddd (Z scale factor)*
BL:XEXTRUDE *Nwwwddd (X component of block extrusion direction)*
BL:YEXTRUDE *Nwwwddd (Y component of block extrusion direction)*
BL:ZEXTRUDE *Nwwwddd (Z component of block extrusion direction)*
*numeric Nwwwddd (Numeric attribute tag)*
*character Cwww*000 *(Character attribute tag)*
The template file can include any or all of the BL:*xxxxxxx* field names listed, but must include at least one attribute tag field. The attribute tag fields determine which attributes, hence which blocks, are included in the attribute extraction file. If a block contains some, but not all, of the specified attributes, the values for the absent ones are filled with blanks or zeros, depending on whether the field is a character field or a numeric field.

Comments should not be included in an attribute template file.

The illustration and table show an example of the type of information you're likely to extract, including block name, manufacturer, model number, and cost.

| Field | (C)haracter or (N)umeric data | Maximum field length | Decimal places |
|---|---|---|---|
| Block name | C | 040 | 000 |
| Manufacturer | C | 006 | 000 |
| Model | C | 015 | 000 |
| Cost | N | 006 | 002 |

You can create any number of template files, depending on how you'll use the data. Each line of a template file specifies one field to be written in the attribute extraction file.

Follow these additional guidelines:

- Be sure to place a space between the attribute tag and the character or numeric data. Use SPACEBAR , not TAB , to enter the space.
- Press ENTER at the end of each line, including the last line.
- Each attribute extraction template file must include at least one attribute tag field, but the same field can appear only once in the file.

The following is a sample template file.

BL:NAME C008000 *(Block name, 8 characters)*

BL:X N007001 *(X coordinate, format nnnnnn.d)*

BL:Y N007001 *(Y coordinate, format nnnnnn.d)*

SUPPLIER C016000 (*Manufacturer's name, 16 characters)*

MODEL C009000 *(Model number, 9 characters)*

PRICE N009002 *(Unit price, format nnnnnnnn.dd)*

---

**NOTE** The format code for a numeric field includes the decimal point in the total field width. For example, the minimum field width to accommodate the number 249.95 would be 6 and would be represented as N006002. Character fields do not use the last three digits of the format code.

---

#### Create an Attribute Extraction File

After creating a template file, you can extract the attribute information using one of the following formats:

- Comma-delimited format (CDF)
- Space-delimited format (SDF)
- Drawing interchange format (DXF)

The CDF format produces a file containing one record for each block reference in a drawing. A comma separates the fields of each record, and single quotation marks enclose the character fields. Some database applications can read this format directly.

The SDF format also produces a file containing one record for each block reference in a drawing. The fields of each record have a fixed width and employ neither field separators nor character-string delimiters. The dBASE III Copy . . . SDF operation also produces SDI-format files. The Append From... SDF

operation can read a file in dBASE IV format, which user programs written in FORTRAN can easily process.

DXF produces a subset of the drawing interchange format containing only block reference, attribute, and end-of-sequence objects. This option requires no attribute extraction template. The file extension *.dxx* distinguishes an extraction file in DXF format from normal DXF files.

#### Use the Attribute Extraction File

The attribute extraction file lists values and other information for the attribute tags you specified in the template file.

If you specified a CDF format using the sample template, the output might appear as follows:

'DESK', 120.0, 49.5, 'ACME INDUST.', '51-793W', 379.95
'CHAIR', 122.0, 47.0, 'ACME INDUST.', '34-902A', 199.95
'DESK', -77.2, 40.0, 'TOP DRAWER INC.', 'X-52-44',249.95
By default, character fields are enclosed with single quotes (apostrophes). The default field delimeter is a comma. The following two template records can be used to override these defaults:

C:QUOTE c (Character string delimeter)
C:DELIM c (Field delimeter)
The first nonblank character following the C:QUOTE or C:DELIM field name becomes the respective delimiter character. For example, if you want to enclose character strings with double quotes, include the following line in your attribute extraction template file:

C:QUOTE "
The quote delimiter must not be set to a character that can appear in a character field. Similarly, the field delimiter must not be set to a character that can appear in a numeric field.

If you specified an SDF format using the sample template, the file might be similar to the following example.

| (NAME) | (X) | (Y) | (SUPPLIER) | (MODEL) | (PRICE) |
|---|---|---|---|---|---|
| DESK | 120.0 | 49.5 | ACME 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 |

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 555 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.

# 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  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  to create a separate drawing file from the library drawing block. Then, use INSERT to overwrite the block definition in the drawing that uses the block.

---

**NOTE** Block descriptions are stripped off when using INSERT. Use the Clipboard to copy and paste a block description displayed in the Block Definition dialog box from one block definition to another.

---

### Modify the Description of a Block

To modify the DesignCenter description of a block definition, use . 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 551

"Modify Dynamic Block Definitions" on page 538

**To update a block definition that originated from a drawing file**

1. On the Tools menu, click 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 alert box, click Yes to overwrite the existing block definition.
7. Press ESC to exit the command.

**To modify a block description**

1. On the Modify menu, click 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. A message box displays the warning "Block name is already defined. Do you want to re-define it?" Click Yes to redefine the block.

# Modify Block Attributes

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  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  to update attribute properties in block references to match their block definition.

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 modify attribute properties and values in a block reference using the Enhanced Attribute Editor. For example, you can change options that determine how attribute text appears in the block, and you can modify

properties that set the layer, linetype, color, lineweight, or plot style of the attribute.

**See also:**

"Define Block Attributes" on page 552
"Modify a Block Definition" on page 561

### To edit attributes assigned to a block definition

1. On the Modify menu, click Object ➤ Attribute ➤ Block Attribute Manager.
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.

Modify II toolbar

### To specify whether changes are applied to existing block references

1. On the Modify menu, click Object ➤ Attribute ➤ Block Attribute Manager.
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.

Modify II toolbar

**To highlight duplicate attribute tags in a block**

1. On the Modify menu, click Object ➤ Attribute ➤ Block Attribute Manager.
2. In the Block Attribute Manager, click Settings.
3. In the Settings dialog box, select Emphasize Duplicate Tags.
4. Click OK.

Modify II toolbar

**To change the prompt order for attribute values**

1. On the Modify menu, click Object ➤ Attribute ➤ Block Attribute Manager.
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).

Modify II toolbar

**To remove an attribute from a block definition and all block references**

1. On the Modify menu, click Object ➤ Attribute ➤ Block Attribute Manager.
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.

Modify II toolbar

**To update existing block references with attributes you have modified**

1 On the Modify menu, click Object ➤ Attribute ➤ Block Attribute Manager.

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.

Modify II toolbar

**To update the attributes in block references for a selected block definition**

1 On the Modify II toolbar, click 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.

Modify II toolbar

**Alternate**

1. On the Modify menu, click Object ➤ Attribute ➤ Single.
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.

Modify II toolbar

# Change the Color and Linetype in a Block

Blocks containing objects with floating properties inherit their color and linetype from the layer on which they are inserted. Depending on how the objects in the block were created, blocks can also inherit floating color and linetype properties from the current explicit color and linetype that you set to override the layer settings.

If a block was not created using objects with floating color and linetype properties, the only way to change these properties is to redefine the block.

**See also:**

"Control the Color and Linetype Properties in Blocks" on page 542

**To change the layer of an object**

1. Select the objects whose layer you want to change.
2. On the Layers toolbar, click the Layer control.

3 Select the layer that you want to assign to the objects.

**To change the color assigned to a layer**

1 On the Layers toolbar, click the Layer Properties Manager button.

2 In the Layer Properties Manager, click the color you want to change.

3 In the Select Color dialog box, use one of the following methods:

- On the Index tab, click a color and then click OK.
- On the Index tab, enter the ACI color number (1-255) or name in the Color box, and then click OK.
- On the True Color tab, select the HSL color model in the Color Model option and specify a color by entering a color value in the Color box or by specifying values in the Hue, Saturation, and Luminance boxes, and then click OK.
- On the Color Books tab, select a color book from the Color Book box, select a color by navigating the color book (using the up and down arrows) and clicking on a color chip, and then click OK.

4 Click OK.

Layers toolbar

**To change the color of an object, overriding the layer's color**

1 On the Standard toolbar, click Properties.

2 Select the objects whose color you want to change.

3 In the Properties palette, select Color.

An arrow is displayed in the right column.

4 Click the arrow and select a color from the list.

Standard toolbar

**To change the linetype assigned to a layer**

1. On the Layers toolbar, click the Layer Properties Manager button.
2. In the Layer Properties Manager, click Load and click one or more linetypes to load, and then click OK.

   You can hold down CTRL to select several linetypes or SHIFT to select a range of linetypes.
3. Select a layer from the list and then click Details to expand the dialog box.
4. Select a linetype from the Linetype list.
5. Click OK to close the dialog box.

   You can click Layer on the Format menu to open the Layer Properties Manager.

Layers toolbar

**To change the linetype of an object, overriding the layer's linetype**

1. Select the objects whose linetype you want to change.
2. On the Properties toolbar, click the Linetype control.
3. Click the linetype that you want to assign to the objects.

**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.

# Disassemble a Block Reference (Explode)

If you need to modify one or more objects within a block separately, you can disassemble, or explode, the block reference into its component objects. After making the changes, you can

- Create a new block definition

- Redefine the existing block definition
- Leave the component objects uncombined for other uses

You can automatically explode block references as you insert them by selecting the Explode option in the Insert dialog box.

**To explode a block reference**

1. On the Modify menu, click Explode.
2. Select the block to explode and press ENTER.

   The block reference is disassembled into its component objects; however, the block definition still exists in the drawing for insertion later.

Modify toolbar

# 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  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 452

**To remove a block definition**

1. On the File menu, click Drawing Utilities ➤ Purge.

   The Purge dialog box displays a tree view of named objects that can be purged.
2. To purge blocks, use one of the following methods:

   - To purge all unreferenced blocks, select Blocks. To include nested blocks, select Purge Nested Items.

- To purge specific blocks, double-click Blocks to expand the Block tree view. Select the blocks to be purged.

If the item you want to purge is not listed, select View Items You Cannot Purge.

3. You are prompted to confirm each item in the list. If you do not want to confirm each purge, clear the Confirm Each Item to Be Purged option.
4. Click Purge.

   To confirm the purging of each item, respond to the prompt by choosing Yes or No, or Yes to All if more than one item is selected.
5. Select more items to purge, or click Close.

# Change Existing Objects

# 21

You can select objects, view and edit object properties, and perform general and object-specific editing operations.

## In this chapter


- Select Objects
- Correct Mistakes
- Erase Objects
- Use Windows Cut, Copy, and Paste
- Modify Objects
- Modify Complex Objects
- Modify 3D Solids

# Select Objects

You have a wide range of options when you need to select objects for editing operations.

# Select Objects Individually

At the Select Objects prompt, you can select one or more objects individually.

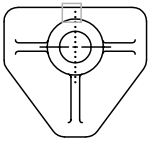
### Use the Pickbox Cursor

When the square pickbox cursor is in position to select an object, the object is highlighted. Click to select the object.

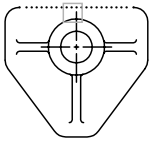
You can control the size of the pickbox in the Options dialog box, Selection tab.

### Select Objects Close Together

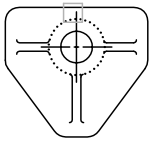
It is difficult to select objects that are close together or lie directly on top of one another. The example shows two lines and a circle that lie within the pickbox.



first selected object



second selected object



third selected object

Hold down CTRL and click to cycle through these objects, one after the other, until the one you want is highlighted. Press ESC to turn off cycling. (Selection previewing is not available when you press CTRL.)

### 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 CTRL. Click as near as possible to the object you want.

Selection previewing is not available when you press CTRL.

2 Keep clicking until the object you want is highlighted.

3 Press ENTER to select the object.

#### To remove selection from objects

- Hold down SHIFT. Click the objects that you want removed from the selection set.

## Select Multiple Objects

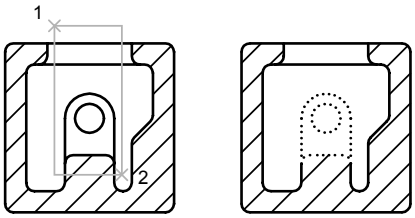
At the Select Objects prompt, you can select many objects at the same time.

#### Specify a Rectangular Selection Area

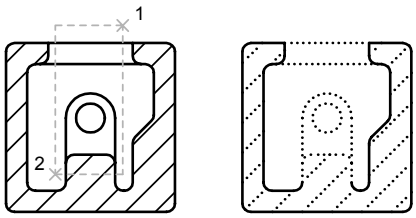
Specify opposite corners to define a rectangular area. The background inside the area changes color and becomes transparent. The direction that you drag your cursor from the first point to the opposite corner determines which objects are selected.

- **Window selection.** Drag your cursor from left to right to select only objects that are entirely enclosed by the rectangular area.

■ **Crossing selection.** Drag your cursor from right to left to select objects that the rectangular window encloses or crosses.




objects selected using window selection box

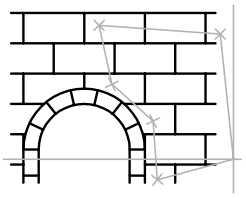



objects selected using crossing selection box

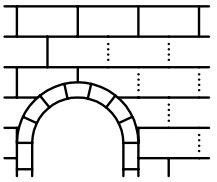
With a window selection, usually the entire object must be contained in the rectangular selection area. However, if an object with a noncontinuous (dashed) linetype is only partially visible in the viewport and all the visible vectors of the linetype can be enclosed within the selection window, the entire object is selected.

#### Specify an Irregularly Shaped Selection Area

Specify points to define an irregularly shaped area. Use window polygon selection to select objects entirely enclosed by the selection area. Use crossing polygon selection to select objects enclosed *or* crossed by the selection area.



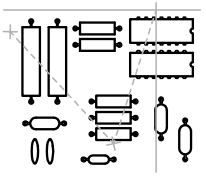
window polygon



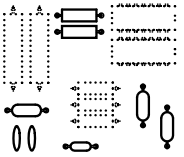
result

### Specify a Selection Fence

In a complex drawing, use a selection fence. A selection fence looks like a polyline and selects only the objects it passes through. The circuit board illustration shows a fence selecting several components.



fence selection



selected objects highlighted

### Use Other Selection Options

You can see all selection options by entering **?** at the Select Objects prompt. For a description of each of the selection options, see .

### 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).

# 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 ), use object snaps to specify points on objects on locked layers, and change the draw order of objects on locked layers.

**To lock or unlock a layer**

1. On the Layers toolbar, click the Layer Properties Manager button.
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.

Layers toolbar

# Filter Selection Sets

You can use object properties or object types to include objects in a selection set, or to exclude them.

Using either Quick Select () from the Properties palette or the Object Selection Filters dialog box (), you can filter selection sets by property (such as color) and by object type. For example, you can select all of the red circles in a drawing without selecting any other object, or you can select all objects except the red circles.

With Quick Select, you can quickly define a selection set based on filtering criteria that you specify, and if an Autodesk or a third-party application was used to add a feature classification to an object, you can select objects by classification property. With object selection filters, you can name and save filters for future use.

With either Quick Select or object selection filters, if you want to filter your selection set based on color, linetype, or lineweight, first consider whether these properties are set to BYLAYER for any objects in your drawing. For example, an object may appear red because its color is set to BYLAYER and the layer color is red.

**See also:**

"Customize Object Selection" on page 582
"Work with Layers" on page 268

**To create a selection set using Quick Select**

In the following example, you use Quick Select to select the red objects in a drawing.

1. Click Tools menu ➤ Quick Select.
2. In the Quick Select dialog box, under Apply To, select Entire Drawing.
3. Under Object Type, select Multiple.
4. Under Properties, select Color.
5. Under Operator, select Equals.
6. Under Value, select Red.
7. Under How to Apply, select Include in New Selection Set.
8. Click OK.

   All red objects in the drawing are selected and the Quick Select dialog box closes. Objects that are set to BYLAYER and are red because the layer color is red are not included in the selection set.

---

**NOTE** If an application such as Autodesk Map was used to add a feature classification to an object, and the associated classification (XML) file is present, you can select objects by classification property. Specifically, you can select a classification in the Object Type box and a property in the Properties box.

---

**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. With several objects selected, click Tools menu ➤ Quick Select.
2. In the Quick Select dialog box, under Apply To, select Current Selection.
3. Under Object Type, select Circle.
4. Under Properties, select Radius.
5. Under Operator, select Greater Than.
6. Under Value, enter **1**.
7. Under How to Apply, select Exclude from New Selection Set.
8. 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 Tools menu ➤ 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.

# 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.

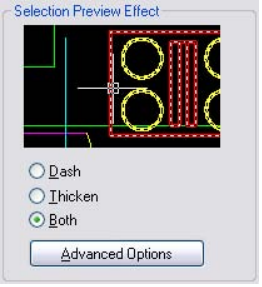
### Select Objects First

You can use one of two methods to select objects before starting a command:

- Use the  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 , , or . 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  system variable is set to 0, selection previewing of objects is not available.

### Control the Appearance of Selected Objects

By default, selected objects are displayed with dashed lines. You can increase program speed by setting the  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. ()
- Press SHIFT to append objects to the selection set. ()
- Click and drag to create a selection window. Otherwise you must click twice to define the corners of a selection window. ()
- Start Window or Crossing selection automatically when you click an empty space. Otherwise, you must enter **c** or **w** to specify window crossing selection. ()
- Change the size of the 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 ➤ Selection tab.
2. 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". ( system variable)
- **Crossing Selection Color**. Select a color, or click Select Color to display the Select Color Dialog Box. ( 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. ( system variable)

4 Click OK to exit each dialog box.

# Group Objects

A group is a saved set of objects that you can select and edit together or separately as needed. Groups provide an easy way to combine drawing elements that you need to manipulate as a unit.

**See also:**

"Create and Use Blocks (Symbols)" on page 451

## 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.

**NOTE** The buttons in the Group Manager become available depending on what you have selected. For example, the Ungroup button is available only when one or more group names are selected in the Group Manager.

To close the Group Manager, undock it and click the Close button at the upper-right corner.

## 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.

- **Unnamed groups.** Create an unnamed group by selecting the objects you want to group and then choosing Group from the Tool menu or choosing 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 a group**

1. At the Command prompt, enter **group**.
2. In the Object Grouping dialog box, under Group Identification, enter a group name and a description.
3. In the Create Group area, click New.

   The dialog box closes temporarily.
4. Select objects and press ENTER .
5. Click OK.

## Select Objects in Groups

There are several methods for choosing a group, including selecting the group by name or selecting one of the members of the group.

You can select groups by name at the Select Objects prompt. If the system variable is set to 1 or 3 and you select any member of a selectable group, all group members that meet the selection criteria are selected. You can also toggle group selection on and off by pressing CTRL + H or SHIFT + CTRL + A .

All members of selectable groups are also selected when you use object selection cycling (for example, if you want to select an object that lies directly behind another object). Selecting an object that is a member of more than one selectable group selects all the members of all the groups that contain that object. To select groups for editing with grips, use the pointing device to select the group at the Command prompt.

**To toggle group selection**

- At any time, toggle group selection on and off by pressing either CTRL + H or SHIFT + CTRL + A .

## Edit Groups

You can add or remove group members and rename groups. You can also perform editing operations, such as copy, mirror, and array, on groups.

Erasing a group member deletes that object from the group definition. When a group member is included in a deleted block, the object is deleted from the drawing and also from the group. If deleting an object or removing it from a group leaves the group empty, the group remains defined. You can remove the group definition by exploding the group. Exploding a group deletes it

from a drawing; however, objects that were part of the group remain in the drawing.

You can alter the group's member order (the order in which the objects were selected), its description, and whether it can be selected. You can reorder group members in two ways: either change the numerical position of individual members or ranges of group members, or reverse the order of all members. The first object in each group is number 0, not number 1.

You can use the Group Manager to add or remove group members. Erasing an object that is a member of a group deletes that object from the group definition. You can also use the Group Manager to revise a group's name or description.

Exploding an object within a group does not automatically add the resulting components to the group.

You can delete a group description by using the Ungroup options on the menu or toolbar in the Group Manager.

---

**NOTE** Exploding an object in a group removes the exploded components from the group definition. See "Disassociate Compound Objects (Explode)" on page 646.

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#### To delete a named group

1. At the Command prompt, enter **group**.
2. In the Object Grouping dialog box, select the group name from the list of groups.
3. Under Change Group, select Explode.
4. Click OK.

   The group is deleted.

#### To reorder group members

1. At the Command prompt, enter **group**.
2. In the Object Grouping dialog box, under Change Group, click Re-Order.
3. In the Order Group dialog box, under Group Name, select the group to reorder.
4. To view the current order of this group, click Highlight.
5. In the Object Grouping dialog box, click Next or Previous to view the objects. Click OK when you have finished viewing the order of the objects.

6 In the Order Group dialog box, under Remove From Position, enter an object number.

7 Under Enter New Position Number for the Object, enter a new position.

8 Under Number of Objects, enter the object number or range of numbers to reorder. Click Re-Order.

9 Click OK to close each dialog box.

# 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  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 system variable is set to combine consecutive pan and zoom commands into a single operation when you undo or redo.

---

### Undo Several Actions at Once

Use the Mark option of  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 undo actions you've defined 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 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 592.

### Cancel a Command

You can cancel a command without completing it by pressing ESC. To change the cancel key assignment, clear the Windows Standard Accelerator Keys option in the Options dialog box, User Preferences tab.

#### To undo the most recent action

■ Click Edit menu ➤ Undo.

Standard toolbar

#### 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.

Standard toolbar

#### 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.

Standard toolbar

#### 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.

Standard toolbar

# Erase Objects

You can erase objects from your drawing using several methods.

- Erase them with .
- Select them and then cut them to the Clipboard with CTRL+X.
- Select them and press DELETE.

You can restore accidentally erased objects using . The  command restores all objects deleted by the most recent use of ERASE, , or .

### 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 .
- To remove stray pixels, use .

### Remove Unused Definitions and Styles

You can remove unused *named objects*, including block definitions, dimension styles, layers, linetypes, and text styles with .

**See also:**

"Correct Mistakes" on page 590

### To erase an object

1. Click Modify menu ➤ 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.

Modify toolbar

### 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 Edit menu ➤ 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 File menu ➤ Drawing Utilities ➤ Purge.

The Purge dialog box displays a tree view of object types with items that can be purged.

2 To purge unreferenced linetypes, use one of the following methods:

- To purge all unreferenced linetypes, select Linetypes.
- To purge specific linetypes, double-click Linetypes to expand the tree view. Then select the linetypes to be purged.

If the item you want to purge is not listed, select View Items You Cannot Purge.

3 You are prompted to confirm each item in the list. If you do not want to confirm each purge, clear the Confirm Each Item to Be Purged option.

4 Click Purge.

To confirm the purging of each item, respond to the prompt by choosing Yes or No, or Yes to All if more than one item is selected.

5 Click Close.

# Use Windows Cut, Copy, and Paste

When you want to use objects from a drawing file in another application, you can cut or copy these objects to the Clipboard and then paste them from the Clipboard into the other application.

### Cut Objects

Cutting deletes selected objects from the drawing and stores them on the Clipboard. The objects are now available to be pasted into other Microsoft® Windows® documents.

### Copy Objects

You can use the Clipboard to copy part or all of a drawing into a document created by another application. The objects are copied in vector format, which retains the high resolution in other applications. These objects are stored in WMF (Windows metafile) format in the Clipboard. The information stored in the Clipboard can then be embedded in the other document. Updating the original drawing does not update the copy embedded in the other application.

### Paste Objects

Applications use different internal formats to store Clipboard information. When you copy objects to the Clipboard, information is stored in all available formats. When you paste the Clipboard contents into a drawing, the format that retains the most information is used. However, you can override this setting and convert pasted information to AutoCAD format.

Because it is the easiest format to edit, the AutoCAD format is the preferred format for copying objects to and from AutoCAD. It retains all relevant object information, including block references and 3D aspects.

The Windows metafile (picture) format contains screen vector information, and files can be scaled and printed without losing resolution. Use this format to paste objects into Windows applications that support WMF files. Metafiles

pasted into AutoCAD are of higher resolution than bitmapped images (BMP files) but are not as easily manipulated as AutoCAD objects. Bitmapped images are raster images consisting of a pattern of pixels and are commonly used by paint applications.

The color of the object doesn't change when copied to the Clipboard. For example, white objects pasted onto a white background won't be visible. Use the  and  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 . If you convert pasted information to AutoCAD format, the object is inserted as a block reference. To edit the pasted information, explode the block reference into its component objects. When you convert a Windows metafile stored on the Clipboard to AutoCAD format, you may lose some scaling precision. To retain proper scaling, save objects in the original drawing as a block (), and then insert them into AutoCAD using .

#### To cut objects to the Clipboard

1. Select the objects you want to cut.
2. Click Edit menu ➤ Cut. You can also press  CTRL +  X .

   The objects are available to be pasted into other Windows applications.

#### To copy objects to the Clipboard

1. Select the objects you want to copy.
2. Click Edit menu ➤ Copy. You can also press  CTRL +  C .

#### To paste objects from the Clipboard

■ Click Edit menu ➤ 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 Edit menu ➤ Paste Special.
2. In the Paste Special dialog box, select Paste.
3. From the list of formats, select Picture.
4. Click OK.

# Modify Objects

You can easily modify the size, shape, and location of objects.

**See also:**

**"Work with Custom and Proxy Objects"**

# Choose a Method to Modify Objects

You can easily modify the size, shape, and location of objects. You can

- Enter a command first, and then select the objects to modify.
- Select the objects first, and then enter a command to modify them.
- Double-click an object to display the Properties palette or, in some cases, a dialog box that is specific to that type of object.
- Select and right-click an object to display a shortcut menu with relevant options.

**See also:**

**"Select Objects" on page 574**
**"Display and Change the Properties of Objects" on page 264**
**"Work with Custom and Proxy Objects"**

# 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.

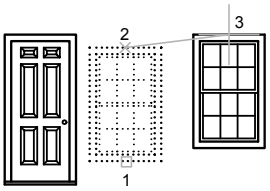
## Move Objects

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. After clicking Move on the Edit menu, 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.

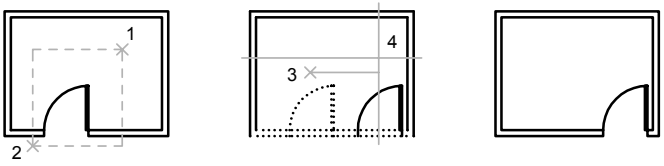
---

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 359

### Use a Stretch-Move

You can also use  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 "Use Grips to Edit Objects" on page 636. 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. See "Embed OLE Objects in Drawings" on page 1003.

### To move an object using two points

1. Click Modify menu ➤ 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.

Modify toolbar

### To move an object using a displacement

1. Click Modify menu ➤ 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.

Modify toolbar

### To move by stretching

1. Click Modify menu ➤ 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.

Modify toolbar

## Rotate Objects

You can rotate objects in your drawing around a specified base point.

To determine the angle of rotation, you can enter an angle value, drag using the cursor, or specify a reference angle to align to an absolute angle.

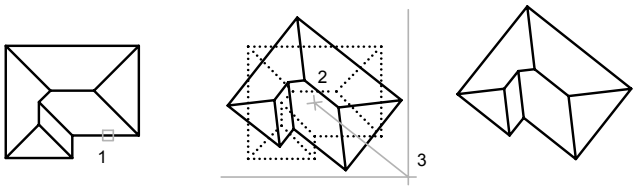
#### Rotate an Object by a Specified Angle

Enter a rotation angle value from 0 to 360 degrees. You can also enter values in radians, grads, or surveyor bearings. Entering a positive angle value rotates the objects counterclockwise or clockwise, depending on the Direction Control setting in the Drawing Units dialog box.

#### Rotate an Object by Dragging

Drag the object around the base point and specify a second point. Use Ortho mode, polar tracking, or object snaps for greater precision.

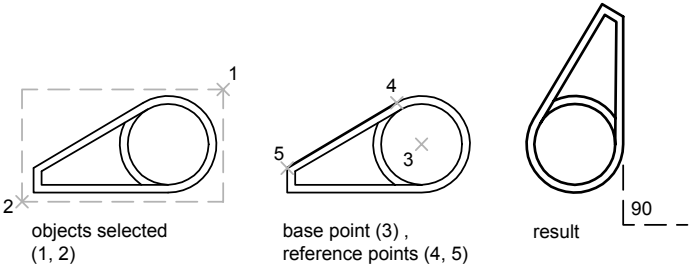
For example, you can rotate the plan view of a house by selecting the objects (1), specifying a base point (2), and specifying an angle of rotation by dragging to another point (3).




### Rotate an Object to an Absolute Angle

With the Reference option, you can rotate an object to align it to an absolute angle.

For example, to rotate the part in the illustration so the diagonal edge rotates to 90 degrees, you select the objects to be rotated (1, 2), specify the base point (3), and enter the Reference option. For the reference angle, specify the two endpoints of the diagonal line (4, 5). For the new angle, enter 90.




### Rotate an Object in 3D

To rotate 3D objects, you can use either ROTATE or ROTATE3D.

- With ROTATE, you can rotate objects around a specified base point. The axis of rotation passes through the base point and is parallel to the Z-axis of the current UCS.
- With ROTATE3D, you can specify the axis of rotation using either two points; an object; the X, Y, or Z axis; or the Z direction of the current view.

**See also:**

"Rotate Views in Layout Viewports" on page 228

### To rotate an object

1. Click Modify menu ➤ 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.

Modify toolbar

### To rotate an object to an absolute angle

1. Click Modify menu ➤ Rotate.
2. Select the objects to rotate.
3. Specify the base point for the rotation.
4. Enter **r** (Reference).
5. Enter a reference angle value or specify two point locations.

   This determines an imaginary line that will be rotated to a new angle.
6. Enter the new angle, or specify a point.

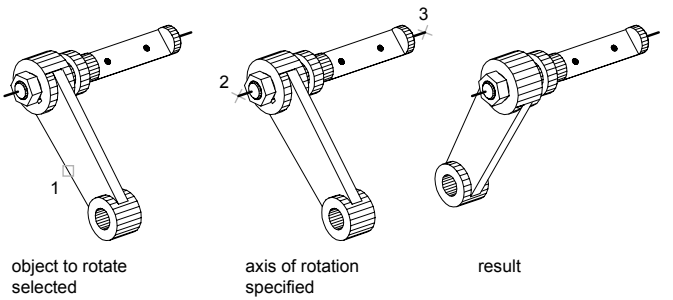
   The value that you enter for the new angle is an absolute angle, not a relative value. Alternatively, if you specify a point, the reference angle will be rotated to that point.

Modify toolbar

**To rotate a 3D object around an axis**

1. Click Modify menu ➤ 3D Operation ➤ Rotate 3D.
2. Select the object to rotate (1).
3. Specify the start point and endpoint of the axis about which the objects are to be rotated (2 and 3).

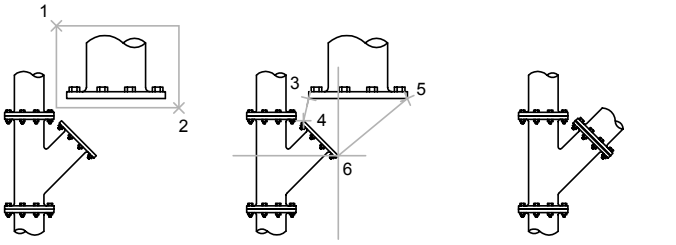
   The positive axis direction is from the start point to the end point, and the rotation follows the right-hand rule (see "Use World and User Coordinate Systems in 3D" on page 326).
4. Specify the angle of rotation.




## Align Objects

You can move, rotate, or tilt an object so that it aligns with another object.

In the following example, align the pieces of piping using a window selection box to select the object to be aligned. Use the Endpoint object snap to align the pieces precisely.




object selected with window selection

source and destination points

result with the scale option used

**To align two objects**

1. Click Modify menu ➤ 3D Operation ➤ Align.
2. Select the objects you want to align.
3. Specify the first source point and then the first destination point.

   If you press ENTER now, the objects are moved from the source point to the destination point.
4. Specify the second source point, and then the second destination point.
5. Specify the third source point, or press ENTER to continue.
6. Specify whether you want to scale objects to the alignment points.

   The objects are aligned (moved and rotated into position), and then scaled. The first destination point is the base point of the scale, the distance between the first and second source points is the reference length, and the distance between the first and second destination points is the new reference length.

# Copy, Offset, or Mirror Objects

You can create duplicates of objects in your drawing that are either identical or similar to selected objects.

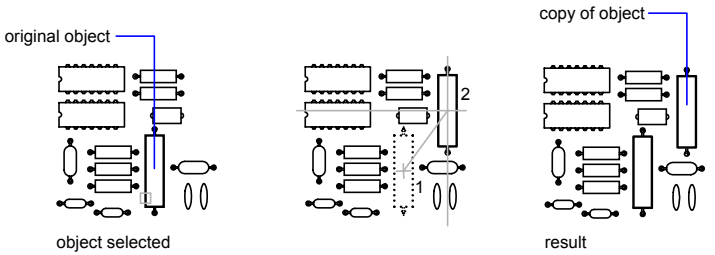
## Copy Objects

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.

### 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. After clicking Copy on the Edit menu, 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

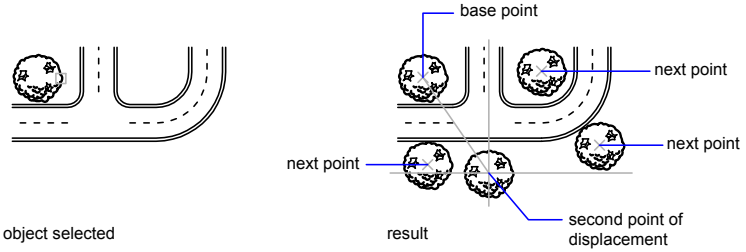
Copy an object using a relative distance by entering coordinate values for the first point and pressing ENTER for the second point. The coordinate values are used as a relative displacement rather than the location of a base point.

**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 359.

### Create Multiple Copies

The  command repeats for convenience. To exit the command, press ENTER.




### Use Alternate Methods

You can use grips to move and copy objects quickly. See "Use Grips to Edit Objects" on page 636. 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. See "Embed OLE Objects in Drawings" on page 1003.

### To copy an object using two points

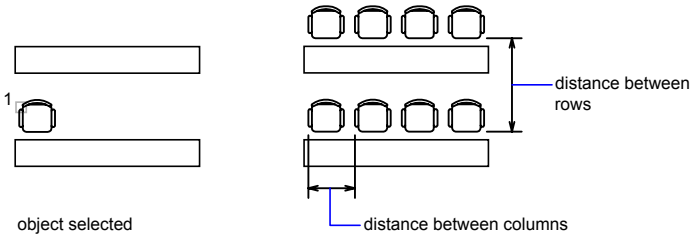
1. Click Modify menu ➤ Copy.
2. Select the objects to copy.
3. Specify the base point.
4. Specify the second point. Press ENTER.

Modify toolbar

## Create an Array of Objects

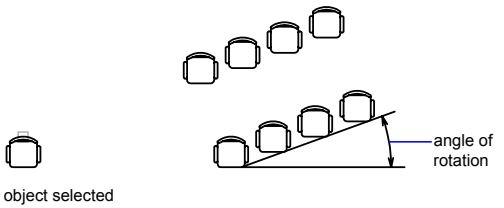
You can create copies of objects in a rectangular or polar (circular) pattern called an array.

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 .




### Create Polar Arrays

When you create a polar array, the array is drawn counterclockwise or clockwise, depending on whether you enter a positive or a negative value for the angle to fill.

The radius of the array is determined by the distance from the specified center point to a *reference* or base point on the last selected object. You can use the

default reference point (usually an arbitrary point that coincides with a snap point), or you can specify a new base point to be used as the reference point.

### Array in 3D

With , you can create a rectangular array or a polar array of objects in 3D. In addition to specifying the number of columns (*X* direction) and rows (*Y* direction), you also specify the number of levels (*Z* direction).

### Limit the Size of Arrays

If you specify a very large number of rows and columns for an array, it may take a long time to create the copies. By default, the number of array elements that can be generated by one command is limited to 100,000. This limit is controlled by the MaxArray setting in the registry.

You can change the limit by setting the MaxArray system registry variable using (**setenv "MaxArray" "n"**) where *n* is a number between 100 and 10000000 (ten million).

---

**NOTE** When changing the value of MaxArray, you must enter MaxArray with the capitalization shown.

---

### To create a rectangular array

1. Click Modify menu ➤ 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.

Modify toolbar

**To create a polar array**

1 Click Modify menu ➤ 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.

Modify toolbar

**To create a 3D rectangular array of objects**

1 Click Modify menu ➤ 3D Operation ➤ 3D Array.

2 Select the object to array (1).

3 Specify Rectangular.

4 Enter the number of rows.

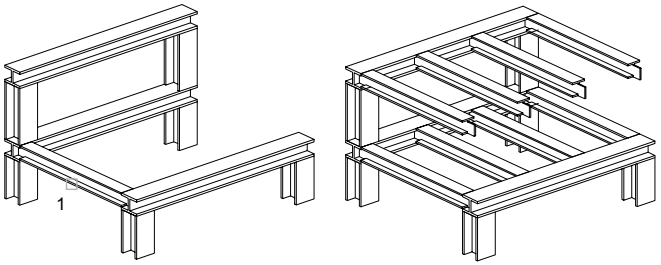
5 Enter the number of columns.

6 Enter the number of levels.

7 Specify the distance between rows.

8 Specify the distance between columns.

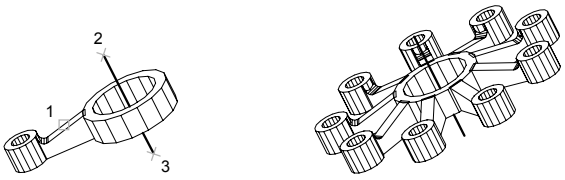
9 Specify the distance between levels.




object to array selected result

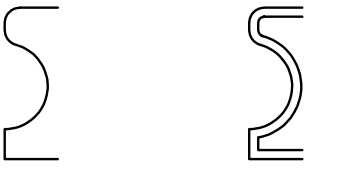
**To create a 3D polar array of objects**

1. Click Modify menu ➤ 3D Operation ➤ 3D Array.
2. Select the object to array (1).
3. Specify Polar.
4. Enter the number of items to array.
5. Specify the angle that the arrayed objects are to fill.
6. Press ENTER to rotate the objects as they are arrayed, or enter **n** to retain their orientation.
7. Specify the start point and endpoint of the axis about which the objects are to be rotated (2 and 3).

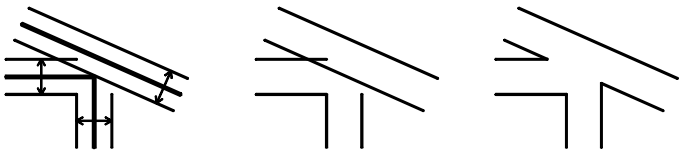



## Offset an Object

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.

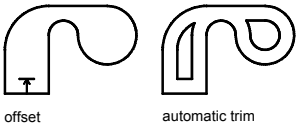



You can offset

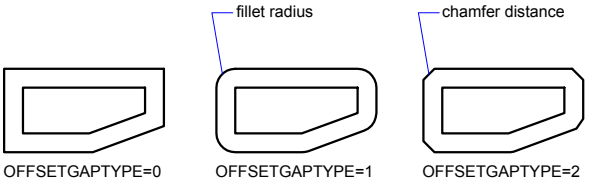
- Lines
- Arcs
- Circles
- Ellipses and elliptical arcs (resulting in an oval-shaped spline)
- 2D polylines
- Construction lines (xlines) and rays
- Splines

### 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.




Closed 2D polylines that are offset to create larger polylines result in potential gaps between segments. The  system variable controls how these potential gaps are closed.




**To offset an object by specifying a distance**

1. Click Modify menu ➤ 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.

Modify toolbar

**To offset an object through a point**

1. Click Modify menu ➤ 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.

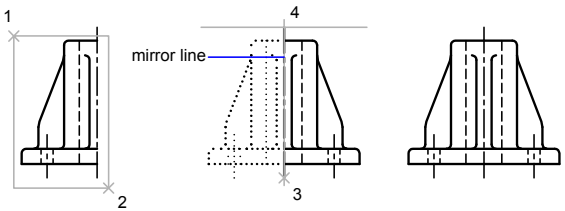
Modify toolbar

## Mirror Objects

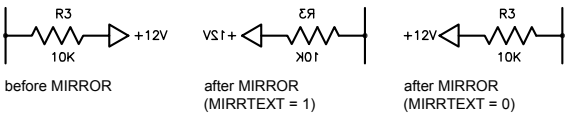
You can flip objects about a specified axis to create a symmetrical mirror image.

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.




When you mirror text, attributes, and attribute definitions, they are reversed or turned upside down in the mirror image. If you want to prevent this, set the  system variable to 0 (off). Text then has the same alignment and justification as before the mirroring.

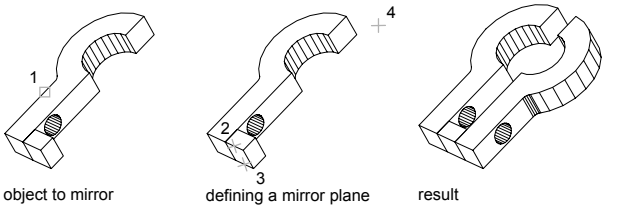



MIRRTEXT affects text that is created with the , , or  commands; attribute definitions; and variable attributes. Text and constant attributes within an inserted block are mirrored as a consequence of mirroring the entire block. These objects are reversed regardless of the MIRRTEXT setting.

### Mirror in 3D

With , you can mirror objects across a specified mirroring plane. The mirroring plane can be one of the following:

- The plane of a planar object
- A plane parallel to the *XY*, *YZ*, or *XZ* plane of the current UCS that passes through a specified point
- A plane defined by three specified points (2, 3, and 4)




### To mirror objects

1. Click Modify menu ➤ 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.

Modify toolbar

### To mirror objects in 3D

1. Click Modify menu ➤ 3D Operation ➤ Mirror 3D.
2. Select the object to mirror.
3. Specify three points to define a mirroring plane.
4. Press ENTER to retain the original objects, or enter **y** to delete them.

# 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.

---

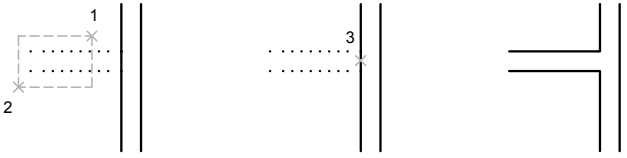
**NOTE** To select cutting edges or boundary edges that include blocks, you can use only the single selection, Crossing, Fence, and Select All options.

---

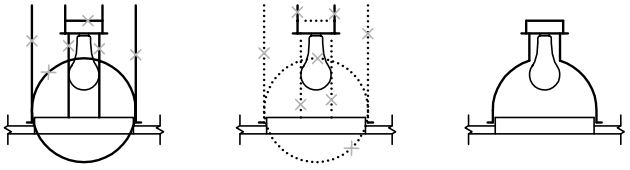
### Trim Objects

You can trim objects so that they end precisely at boundary edges defined by other objects.

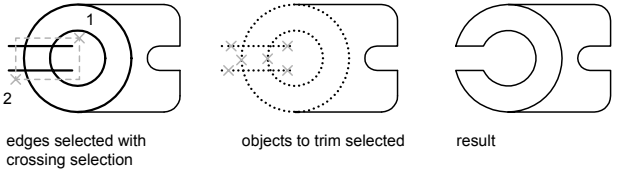
For example, you can clean up the intersection of two walls smoothly by trimming.



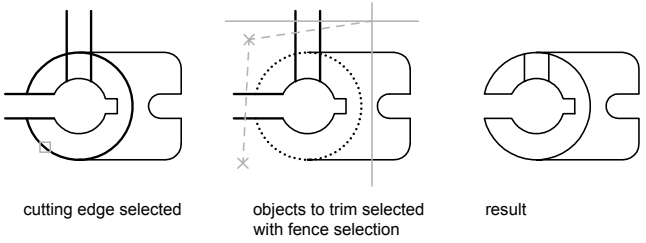

An object can be one of the cutting edges and one of the objects being trimmed. For example, in the illustrated light fixture, the circle is a cutting edge for the construction lines and is also being trimmed.



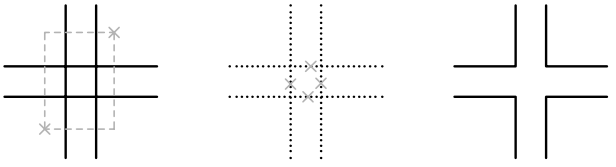

When you trim several objects, the different selection methods can help you choose the current cutting edges and objects to trim. In the following example, the cutting edges are selected using crossing selection.




The following example uses the fence selection method to select a series of objects for trimming.



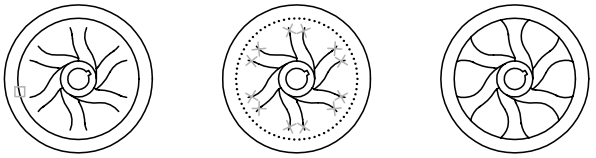

You can trim objects to their nearest intersection with other objects. Instead of selecting cutting edges, you press ENTER. Then, when you select the objects to trim, the nearest displayed objects act as cutting edges. In this example, the walls are trimmed so that they intersect smoothly.




You can extend objects without leaving the  command. Hold down SHIFT and select the objects to be extended.

### Extend Objects

Extending operates the same way as trimming. You can extend objects so they end precisely at boundary edges defined by other objects. In this example, you extend the lines precisely to a circle, which is the boundary edge.

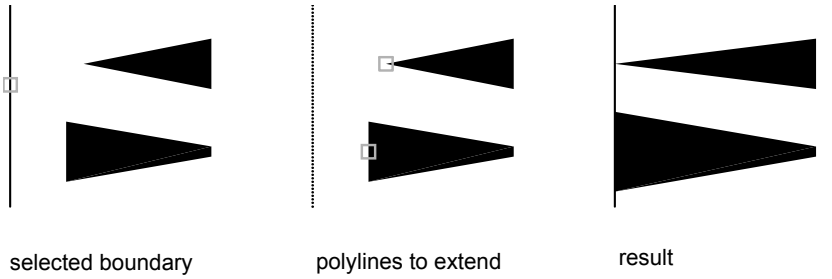



You can trim objects without leaving the EXTEND command. Hold down SHIFT and select the objects to be trimmed.

### Trim and Extend Wide Polylines

2D wide polylines trim and extend at their centerlines. The ends of wide polylines are always square. Trimming a wide polyline at an angle causes portions of the end to extend beyond the cutting edge

If you trim or extend a tapered 2D polyline segment, the width of the extended end is corrected to continue the original taper to the new endpoint. If this correction gives the segment a negative ending width, the ending width is forced to 0.




### Trim and Extend Spline-Fit Polylines

Trimming a spline-fit polyline removes the curve-fit information and changes the spline-fit segments into ordinary polyline segments.

Extending a spline-fit polyline adds a new vertex to the control frame for the polyline.

### Trim or Extend in 3D

You can trim or extend an object to any other object in 3D space, regardless of whether the objects are on the same plane or parallel to the cutting or boundary edges. In the TRIM and EXTEND commands, use the Project and Edge options to select one of three projections for trimming or extending:

- The *XY* plane of the current UCS
- The plane of the current view
- True 3D, which is not a projection

**See also:**

"Break and Join Objects" on page 634

### To extend an object

1. Click Modify menu ➤ 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.

Modify toolbar

**To trim an object**

1. Click Modify menu ➤ Trim.
2. Select the objects to serve as cutting edges.

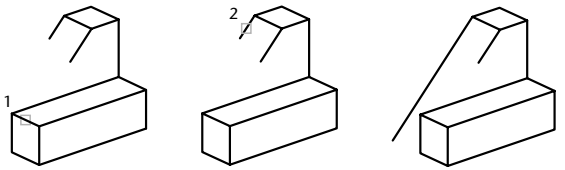
   To select all displayed objects as potential cutting edges, press ENTER without selecting any objects.
3. Select the objects to trim.

Modify toolbar

**To extend objects in 3D wireframe models**

1. Click Modify menu➤ Extend.

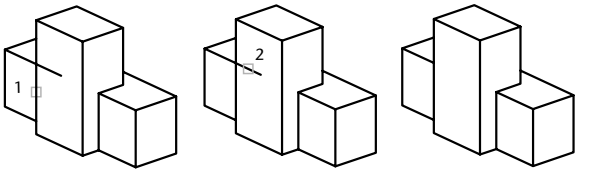
2. Select the boundary edge for extending (1).
3. Enter **e** (Edge).
4. Enter **e** (Extend).
5. Enter **p** (Project).
6. Enter **u** (UCS).
7. Select the object to extend (2).



**To trim in 3D using the current view plane**

1. Click Modify menu ➤ Trim.

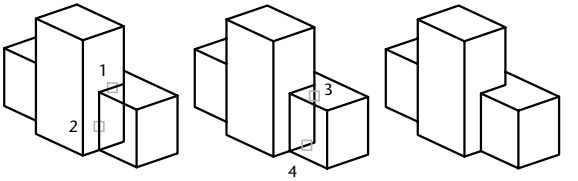
2. Select the cutting edge for trimming (1).
3. Enter **p** (Project).
4. Enter **v** (View).
5. Select the object to trim (2).




**To trim objects in 3D wireframe models**

1. Click Modify menu ➤ Trim.

2. Select the cutting edges to use for trimming (1 and 2).
3. Enter **p** (Project).
4. Enter **n** (None).
5. Select the object to trim (3 and 4).



## 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 , 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

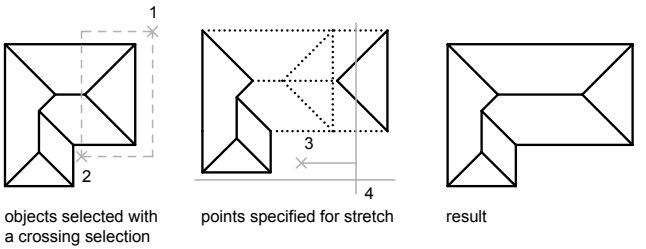
- Drag an object endpoint dynamically
- Specify a new length or angle as a percentage of the total length or angle
- Specify an incremental length or angle measured from an endpoint
- Specify the object's total absolute length or included angle

### 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.



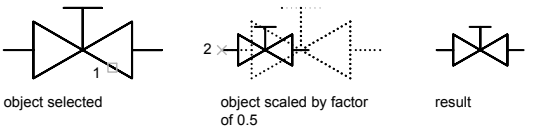

To stretch with precision, use object snaps, grid snaps, and relative coordinate entry.

### Scale Objects Using a Scale Factor

With , 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.




### 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 634

### To stretch an object

1. Click Modify menu ➤ 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.

Modify toolbar

### To scale an object by a scale factor

1. Click Modify menu ➤ 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.

Modify toolbar

### To scale an object by reference

1. Click Modify menu ➤ 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.

Modify toolbar

**To change the length of an object by dragging**

1 Click Modify menu ➤ 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.

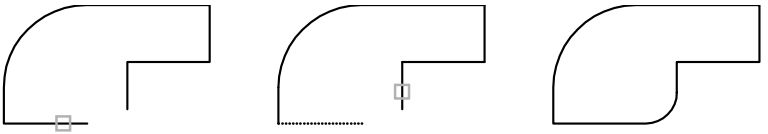
The selected object is lengthened or shortened without changing its location or orientation.

# 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  command.

You can fillet

- Arcs
- Circles
- Ellipses and elliptical arcs
- Lines
- Polylines
- Rays
- Splines
- Xlines
- 3D solids

FILLET can be used to round all corners on a polyline using a single command.

---

**NOTE** Filleting a hatch boundary that was defined from line segments removes hatch associativity. If the hatch boundary was defined from a polyline, associativity is maintained.

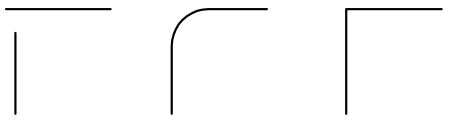
---

If both objects being filleted are on the same layer, the fillet arc is created on that layer. Otherwise, the fillet arc is created on the current layer. The layer affects object properties including color and linetype.

Use the Multiple option to fillet more than one set of objects without leaving the command.

### Set the Fillet Radius

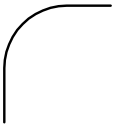
The fillet radius is the radius of the arc that connects filleted objects. Changing the fillet radius affects subsequent fillets. If you set the fillet radius to 0, filleted objects are trimmed or extended until they intersect, but no arc is created.



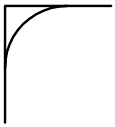

You can hold down SHIFT while selecting the objects to override the current fillet radius with a value of 0.

### Trim and Extend Filleted Objects

You can use the Trim option to specify whether the selected objects are trimmed or extended to the endpoints of the resulting arc or left unchanged.



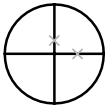
two lines filleted with Trim option set



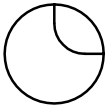
two lines filleted with No Trim option set

### Control the Location of the Fillet

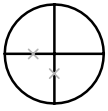
Depending on the locations you specify, more than one possible fillet can exist between the selected objects. Compare the selection locations and resulting fillets in the illustrations.



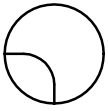
fillet location points selected



result



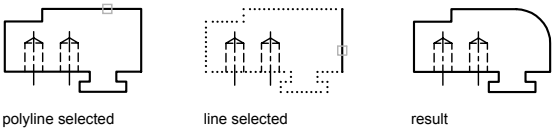
fillet location points selected



result

### Fillet Line and Polyline Combinations

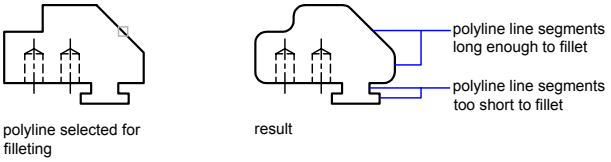
To fillet lines with polylines, each line or its extension must intersect one of the polyline line segments. If the Trim option is on, the filleted objects and the fillet arc join to form a single new polyline.



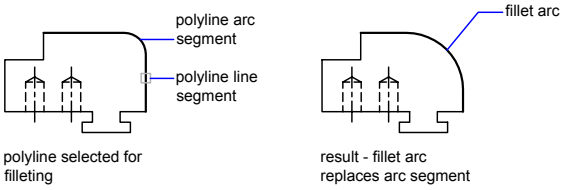

### Fillet an Entire Polyline

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.

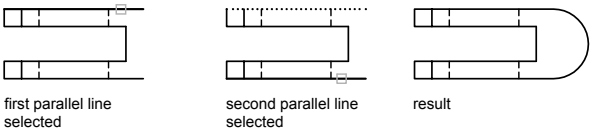



If you set the fillet radius to 0, no fillet arcs are inserted. If two polyline line segments are separated by one arc segment, FILLET removes that arc and extends the lines until they intersect.

### 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 Modify menu ➤ Fillet.
2. Enter **r** (Radius).
3. Enter the fillet radius
4. Select the objects to fillet.

Modify toolbar

### To fillet two line segments

1. Click Modify menu ➤ Fillet.
2. Select the first line.
3. Select the second line.

Modify toolbar

### To fillet without trimming

1. Click Modify menu ➤ Fillet.

2 If necessary, enter **t** (Trim). Enter **n** (No Trim).

3 Select the objects to fillet.

Modify toolbar

#### To fillet an entire polyline

1 Click Modify menu ➤ Fillet.

2 Enter **p** (Polyline).

3 Select the polyline.

Modify toolbar

#### To fillet multiple sets of objects

1 Click Modify menu ➤ 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.

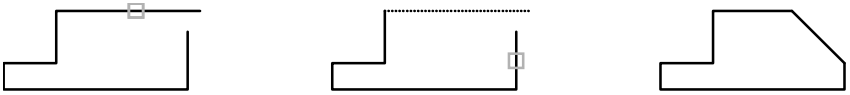
The main prompt is displayed again.

5 Select the first line for the next fillet, or press ENTER or ESC to end the command.

Modify toolbar

## Create Chamfers

A chamfer connects two objects with an angled line. It is usually used to represent a beveled edge on a corner.



first selected line | second selected line | result

You can chamfer

- Lines
- Polylines
- Rays
- Xlines
- 3D solids

CHAMFER can be used to bevel all corners of a polyline using a single command.

---

**NOTE** Chamfering a hatch boundary that was defined from line segments removes hatch associativity. If the hatch boundary was defined from a polyline, associativity is maintained.

---

If both objects being chamfered are on the same layer, the chamfer line is created on that layer. Otherwise, the chamfer line is created on the current layer. The layer affects object properties including color and linetype.

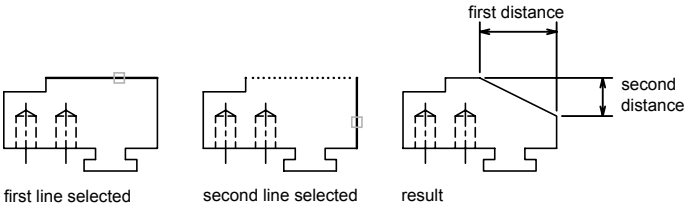
Use the Multiple option to chamfer more than one set of objects without leaving the command.

### Chamfer by Specifying Distances

The chamfer distance is the amount each object is trimmed or extended to meet the chamfer line or to intersect the other. If both chamfer distances are 0, chamfering trims or extends the two objects until they intersect but does not create a chamfer line. You can hold down SHIFT while selecting the objects to override the current chamfer distances with a value of 0.

original objects | chamfer distance zero | chamfer distance nonzero

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.



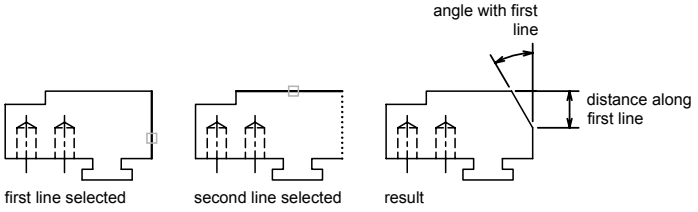

### Trim and Extend Chamfered Objects

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

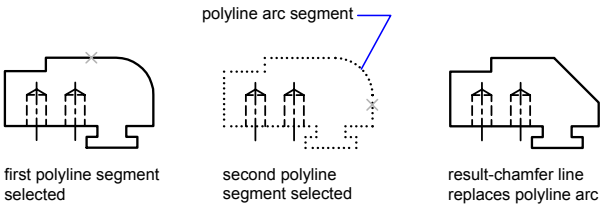
You can chamfer two objects by specifying where on the first selected object the chamfer line starts, and then the angle the chamfer line forms with this object.

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.

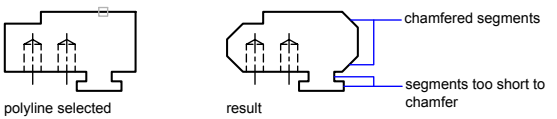



### Chamfer an Entire Polyline

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 Modify menu ➤ Chamfer.
2. Enter **d** (Distances).
3. Enter the first chamfer distance.
4. Enter the second chamfer distance.

5 Select the lines for chamfering.

Modify toolbar

**To chamfer two nonparallel line segments**

1 Click Modify menu ➤ Chamfer.
2 Select the first line.
3 Select the second line.

Modify toolbar

**To chamfer by specifying chamfer length and angle**

1 Click Modify menu ➤ 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.

Modify toolbar

**To chamfer without trimming**

1 Click Modify menu ➤ Chamfer.
2 Enter **t** (Trim Control).
3 Enter **n** (No Trim).
4 Select the objects to chamfer.

Modify toolbar

**To chamfer an entire polyline**

1. Click Modify menu ➤ Chamfer.
2. Enter **p** (Polyline).
3. Select the polyline.

   The polyline is chamfered using the current chamfer method and the default distances.

Modify toolbar

**To chamfer multiple sets of objects**

1. Click Modify menu ➤ 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.

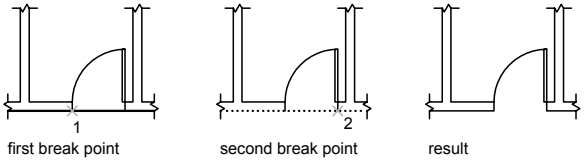
Modify toolbar

## 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  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  to combine similar objects into a single object. You can also create complete circles and ellipses from arcs and elliptical arcs. You can join

- Arcs
- Elliptical arcs
- Lines
- Polylines
- Splines

The object to which you want to join similar objects is called a source object. Objects to be joined must be located in the same plane. Additional restrictions for each type of objects are described in the  command.

---

**NOTE** When joining two or more arcs (or elliptical arcs), the arcs are joined counterclockwise beginning from the source object.

---

**See also:**

"Modify or Join Polylines" on page 647
"Modify Complex Objects" on page 645

### To break an object

1. Click Modify menu ➤ 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.

Modify toolbar

### To join objects

1. Click Modify menu ➤ 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 command.

Modify toolbar

# Use Grips to Edit Objects

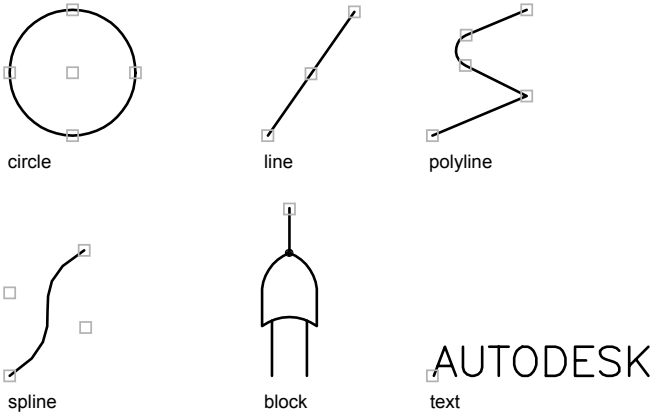
Grips are small squares that are displayed at strategic points on objects that you have selected with a pointing device. You can drag these grips to edit objects directly and quickly.

## Use Grip Modes

You can drag grips to perform any stretch, move, rotate, scale, or mirror operations. You choose the editing operation to be performed by specifying a grip mode.

Grips are small, solid-filled squares that are displayed at strategic points on objects that you have selected with a pointing device. You can drag these grips to stretch, move, rotate, scale, or mirror objects quickly.

When grips are on, you select the objects you want *before* entering a command, and then you manipulate the objects with the pointing device.




To use grips, select a grip to act as the base point for the action. Then select one of the grip modes. You can cycle through these modes by pressing ENTER or SPACEBAR. You can also use shortcut keys or right-click to see all of the modes and options.

You can use multiple grips as the base grips to keep the shape of the object intact between the selected grips. Hold down SHIFT as you select the 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 on the command line 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.

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.

You can limit the display of grips on selected objects. The GRIPOBJLIMIT system variable suppresses the display of grips when the initial selection set includes more than the specified number of objects. If you add objects to the current selection set, the limit does not apply. For example, if GRIPOBJLIMIT is set to 20, you can select 15 objects and then add 25 objects to the selection, and all objects display grips.

### Stretch with Grips

You can stretch an object by moving selected grips to new locations. Grips on text, block references, midpoints of lines, centers of circles, and point objects move the object rather than stretching it. This is an excellent method for moving block references and adjusting dimensions.

### Move with Grips

You can move objects by the grip selected. Selected objects are highlighted and are moved the direction and distance of the next point location you specify.

### Rotate with Grips

You can rotate selected objects around a base point by dragging and specifying a point location. Alternatively, you can enter an angle value. This is an excellent method for rotating block references.

### Scale with Grips

You can scale selected objects relative to a base point. Increase the size of an object by dragging outward from the base grip and specifying a point location, or decrease the size by dragging inward. Alternatively, you can enter a value for relative scaling.

### Mirror with Grips

You can mirror selected objects across a temporary mirror line. Turning Ortho on helps you specify a vertical or horizontal mirror line.

**See also:**

"Use Dynamic Input" on page 335
"Work with Dynamic Blocks in Drawings" on page 548

### To turn on grips

1. On the Tools menu, click Options.
2. In the Options dialog box, Selection tab, select Enable Grips.
3. Click OK.

### To set the color of the unselected grip beneath the cursor

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Selection tab, click the arrow under Hover Grip Color.
3. Select a color, or click Select Color to display the Select Color dialog box.

### To limit the number of objects in the initial selection set that display grips

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Selection tab, enter a number in the Object Selection Limit for Display of Grips box. The maximum is 32,767.

   If you add objects to the current selection set, the limit does not apply.

### To display grip tips for custom objects that support grip tips

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Selection tab, select Enable Grip Tips.

### To cancel grip selection

■ Press ESC.

### To stretch an object using grips

1. Select the object to stretch.
2. Select a base grip on the object.

   The selected grip is highlighted, and Stretch, the default grip mode, is active.
3. Move the pointing device and click.

   The selected object is stretched as the grip moves.

### 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.

### 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.

### 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 Move the pointing device and click.

#### 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.

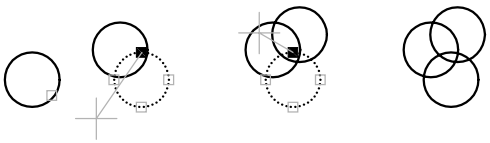
4 Click to specify the second point of the mirror line.

   Turning on Ortho mode is often useful when mirroring objects.

## 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.

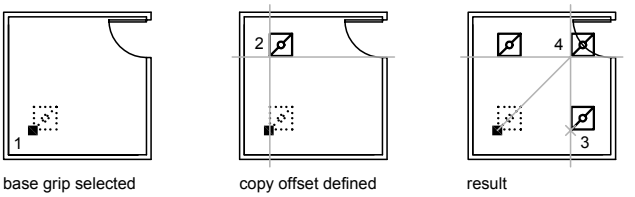


You can also make multiple copies by holding down CTRL as you select the first point. For example, with the Stretch grip mode, you can stretch an object,

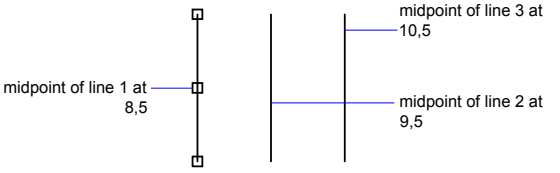
such as a line, and then copy it to any point in the drawing area. Multiple copies continue being made until you turn off grips.

### Define an Offset Snap or a Rotation Snap

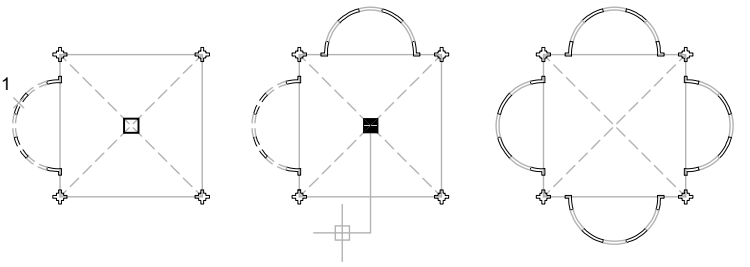
You can place multiple copies at regularly spaced intervals with an offset snap. The offset snap is defined by the distance between an object and the next copy. In the lighting layout below, the first copy of the light fixture symbol is placed at an offset of two units. All subsequent copies are then 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).

   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.

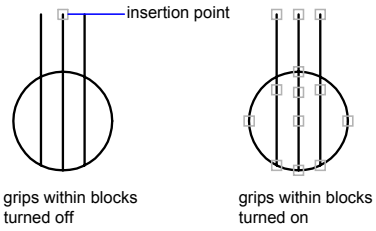
Turning on Ortho mode is often useful when mirroring objects.

5 Turn off grips by pressing ENTER, SPACEBAR, or ESC.

## Control Grips in Blocks

You can specify whether a block displays a single grip or multiple grips.

You can specify whether a selected block reference displays a single grip at its insertion point or displays multiple grips associated with the objects grouped within the block.




**See also:**

"Specify Grips for Dynamic Blocks" on page 510

**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.

# Modify Complex Objects

Additional editing operations are available for complex objects, such as blocks, dimensions, hatches, and polylines.

**See also:**


"Modify 3D Solids" on page 655
"Modify Blocks" on page 561
"Modify Hatches and Solid-Filled Areas" on page 693
"Change Text" on page 742
"Modify Existing Dimensions" on page 824
"Update Fields" on page 730
"Create and Modify Tables" on page 758
"Modify Raster Images and Image Boundaries"


# Disassociate Compound Objects (Explode)

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.

### 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 Modify menu ➤ Explode.
2. Select the objects to be exploded.

For most objects, exploding has no visible effect.

Modify toolbar

# Modify or Join Polylines

Additional editing operations are available for changing the shape of polyline objects. You can also join separate polylines.

You can edit polylines by closing and opening them and by moving, adding, or deleting individual vertices. You can straighten the polyline between any two vertices and toggle the linetype so that a dash appears before and after each vertex. You can set a uniform width for the entire polyline or control the width of each segment. You can also create a linear approximation of a spline from a polyline.

### Joined 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.

### Properties of Modified Polylines

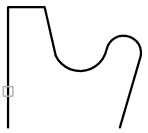
If the properties of several objects being joined into a polyline differ, the resulting polyline inherits the properties of the first object that you selected. If two lines meet a polyline in a Y shape, one of the lines is selected and joined to the polyline. Joining also causes an implicit decurve, with the program discarding the spline information of the original polyline and any polylines being joined to it. Once the joining is completed, you can fit a new spline to the resulting polyline.

### Additional Editing Options for Polylines

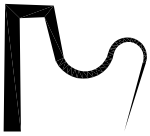
In addition to the general editing operations available for most objects, there are additional options available for editing and joining polylines with .

- **Close.** Creates the closing segment of the polyline, connecting the last segment with the first. The polyline is considered open unless you close it using the Close option.

- **Join.** Adds lines, arcs, or polylines to the end of an open polyline and removes the curve fitting from a curve-fit polyline. For an object to join the polyline, their endpoints must touch.
- **Width.** Specifies a new uniform width for the entire polyline. Use the Width option of the Edit Vertex option to change the starting and ending widths of segments.

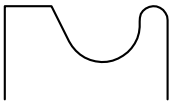


polyline selected



segments with different start and end widths

- **Edit Vertex.** Marks the first vertex of the polyline by drawing an X on the screen. If you have specified a tangent direction for this vertex, an arrow is also drawn in that direction.
- **Fit.** Creates a smooth curve consisting of arcs joining each pair of vertices. The curve passes through all vertices of the polyline and uses any tangent direction you specify.
- **Spline.** Uses the vertices of the selected polyline as the control points, or frame, of a spline-fit polyline. The curve passes through the first and last control points unless the original polyline was closed.



polyline

splined polyline

- **Decurve.** Removes extra vertices inserted by an arc-fit or spline-fit polyline and straightens all segments of the polyline.
- **Ltype Gen.** Generates the linetype in a continuous pattern through the vertices of the polyline. When this option is turned off, the linetype is generated starting and ending with a dash at each vertex.

**See also:**

"Trim or Extend Objects" on page 615
"Break and Join Objects" on page 634

**To modify a polyline**

1. Click Modify menu ➤ Object ➤ Polyline.
2. Select the polyline to modify.

   If the selected object is a line or an arc, the following prompt is displayed:

Object selected is not a polyline.

Do you want it to turn into one? <Y>: *Enter* **y** *or* **n** *, or press ENTER*

   If you enter **y**, the object is converted into a single-segment 2D polyline that you can edit. You can use this operation to join lines and arcs into a polyline. When the  system variable is set to 1, this prompt is suppressed, and the selected object is automatically converted to a polyline.

3. 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, arcs, or polylines.
   - Enter **w** (Width) to specify a new uniform width for the entire polyline.
   - Enter **e** (Edit Vertex) to edit a vertex.
   - Enter **f** (Fit) to create a series of arcs joining each pair of vertices.
   - Enter **s** (Spline) to create an approximation of a spline.
   - Enter **d** (Decurve) to remove extra vertices inserted by a fit or spline curve and to straighten all segments of the polyline.
   - Enter **L** (Ltype Gen) to generate the linetype in a continuous pattern through the vertices of the polyline.
   - Enter **u** (Undo) to reverse actions back to the start of PEDIT.
4. Enter **x** (Exit) to end the command.

Modify II toolbar

**To taper the width of individual polyline segments**

1. Click Modify menu ➤ Object ➤ 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 vertexes.

8 Enter **x** (Exit) again to end the command.

Modify II toolbar

# Modify Splines

Additional editing options are available for changing the shape of spline objects.

In addition to the general editing operations available for most objects, there are additional options available for editing splines with .

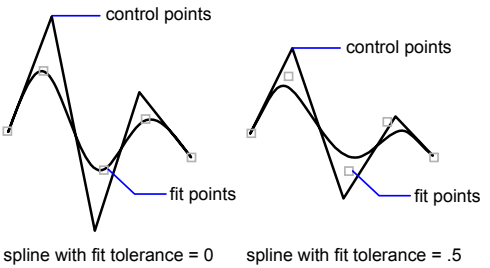
- **Fit Data.** Edits the fit point data that defines the spline, including changing the tolerance.
- **Close.** Changes an open spline into a continuous, closed loop.
- **Move Vertex.** Moves a fit point to a new location.
- **Refine.** Modifies a spline definition by adding and weighting control points and elevating the order of the spline.
- **Reverse.** Changes the direction of the spline.

You also can change the tolerance of the spline. Tolerance refers to how closely the spline fits the set of fit points you specify. The lower the tolerance, the more closely the spline fits the points.

### Edit Splines with Grips

When you select a spline, grips are displayed on its fit points (the  system variable must be set to 1). You can use grips to modify the shape and location of the spline.

After certain operations, fit points are discarded and grips are displayed on control points instead. These operations include trimming the spline, moving the control points, and purging the fit data. If the spline's control frame is turned on (the  system variable is set to 1), grips are displayed on both the spline's control points and its fit points, when available.



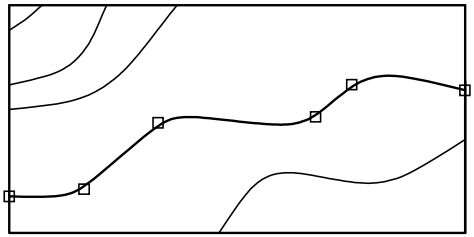

spline with fit tolerance = 0 spline with fit tolerance = .5

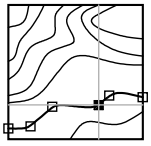
You can delete fit points of a spline, add fit points for greater accuracy, or move fit points to alter the shape of a spline. You can open or close a spline and edit the spline start and end tangents. Spline direction is reversible. You can change the *tolerance* of the spline also. Tolerance refers to how closely the spline fits the set of fit points you specify. The lower the tolerance, the more closely the spline fits the points.

### Refine the Shape of a Spline

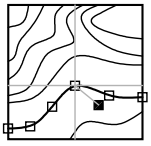
You can refine a spline by increasing the number of control points in one portion of the spline or by changing the weight of specific control points. Increasing the weight of a control point pulls the spline more towards that point. You can also refine a spline by changing its order. A spline's order is the degree of the spline polynomial + 1. A cubic spline, for example, has order 4. The higher a spline's order, the more control points it has.

Consider the following example. You have created a spline to represent a geographic contour. Grips are turned on, and you need to move the fourth fit point to increase accuracy. When you select the spline, grips appear at the control points. If you created the spline by fitting it through a set of points, and you haven't purged this information using the Purge option of the SPLINEDIT command, and you select the Fit Data option, grips appear at the fit points on the selected spline instead of at the control points.

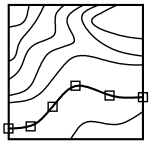




fourth fit point selected



fit point moved



result

**See also:**

"Break and Join Objects" on page 634

**To edit a spline**

1. Click Modify menu ➤ Object ➤ Spline.
2. Select the spline to modify.
3. Edit the spline by entering one or more of the following options:
   - Enter **f** (Fit) to edit the fit data that defines the spline.
   - Enter **c** (Close) to change an open spline into a continuous, closed loop.
   - Enter **m** (Move Vertex) to move a fit point to a new location.
   - Enter **r** (Refine) to modify a spline definition by adding and weighting control points and elevating the order of the spline.
   - Enter **e** (Reverse) to reverse the direction of the spline.
   - Enter **u** (Undo) to cancel the last editing action.
4. Enter **x** (Exit) to end the command.

Modify II toolbar

# Modify Multilines

Multiline objects are composed of 1 to 16 parallel lines, called *elements*. To modify multilines or their elements, you can use common editing commands, a multiline editing command, and multiline styles.

Special multiline editing features are available with the command including the following:

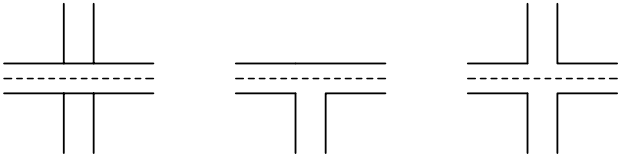
- Add or delete a vertex
- Control the visibility of corner joints
- Control the style of intersection with other multilines
- Open or close gaps in a multiline object

### Add and Delete Multiline Vertices

You can add or delete any vertex in a multiline.

### Edit Multiline Intersections

If you have two multilines in a drawing, you can control the way they intersect. Multilines can intersect in a cross or a **T** shape, and the crosses or **T** shapes can be closed, open, or merged.



### Edit Multiline Styles

You can use  to edit multiline styles to change the properties of multiline elements or the end caps and background fill of subsequently created multilines.

Multiline styles control the number of line elements in a multiline and the color, linetype, lineweight, and offset of each element. You can also modify the display of joints, end caps, and background fill.

Multiline styles have the following limitations:

- You cannot edit the element and multiline properties of the STANDARD multiline style or any multiline style already used in the drawing.
- To edit an existing multiline style, you must do so *before* you draw any multilines in that style.

---

**NOTE** If you use  to create a multiline style without saving it, and then select another style or create a new style, the first MLSTYLE properties are lost. To maintain the properties, save each multiline style to an MLN file before creating a new one.

---

### Use Common Editing Commands on Multilines

You can use most of the common editing commands on multilines *except*

- BREAK
- CHAMFER
- FILLET
- LENGTHEN
- OFFSET

To perform these operations, first use  to replace the multiline object with separate line objects.

---

**NOTE** If you trim or extend a multiline object, only the first boundary object encountered determines the shape of the end of the multiline. A multiline cannot have a complex boundary at its endpoint.

---

**See also:**

"Draw Multiple-Line Objects" on page 397

**To delete a vertex from a multiline**

1. Click Modify menu ➤ Object ➤ Multiline.
2. In the Multiline Edit Tools dialog box, select Delete Vertex.
3. In the drawing, specify the vertex to delete. Press ENTER.

**To create a closed cross intersection**

1. Click Modify menu ➤ Object ➤ Multiline.
2. In the Multiline Edit Tools dialog box, select Closed Cross.
3. Select the multiline for the foreground.
4. Select the multiline for the background.

   The intersection is modified. You can continue selecting intersecting multilines to modify, or press ENTER to end the command. Press ENTER again to redisplay the Multiline Edit Tools dialog box.

**To edit a multiline style**

1. Click Format menu ➤ Multiline Style.
2. In the Multiline Styles dialog box, select the style name from the list. Click Modify.
3. Click Element Properties.
4. In the Modify Multiline Styles dialog box, change the settings as needed.
5. Click OK.
6. In the Multiline Styles dialog box, click Save to save the changes to the style in the MLN file.
7. Click OK.

# Modify 3D Solids

After creating a 3D solid model, you can use the ShapeManager modeler to change the form and appearance of the model.

## Overview of Modifying 3D Solids

After creating a solid model, you can change its appearance by filleting, chamfering, sectioning, slicing, and separating.

You can also edit faces and edges on your solid model. You can easily remove blends created by  or . You can change the color or copy a face or edge of a solid as a body, region, line, arc, circle, ellipse, or spline object. Imprinting geometry on existing solids creates new faces or merges redundant faces. Offsetting changes the faces relative to the original faces on the solid model, for example, making the diameter of a hole larger or smaller. Separating disjointed composite solids creates 3D solid objects. Shelling creates thin walls with a specified thickness.

# Fillet and Chamfer 3D Solids

You can add rounds and fillets to selected edges of 3D solids.

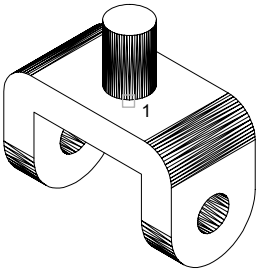
With , you can add rounds and fillets to selected 3D solids. The default method is specifying the fillet radius and then selecting the edges to fillet. Other methods specify individual measurements for each filleted edge and fillet a tangential series of edges.

Similarly, with , you can bevel the edges along adjoining faces of selected 3D solids.

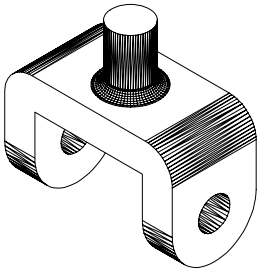
**To fillet a solid object**

1. Click Modify menu ➤ Fillet.

2. Select the edge of the solid to fillet (1).
3. Specify the fillet radius.
4. Select additional edges or press  ENTER  to fillet.




edge to fillet selected



result

**To chamfer a solid object**

1. Click Modify menu ➤ Chamfer.

2. Select the edge of the base surface to chamfer (1).

   One of two surfaces adjacent to the selected edge is highlighted.

3. To select a different surface, enter **n** (Next), or press ENTER to use the current surface.

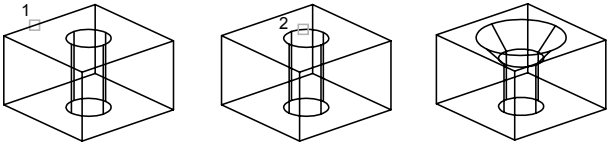
4. Specify the base surface distance.

   The base surface distance is measured from the selected edge to a point on the base surface. The other surface distance is measured from the selected edge to a point on the adjacent surface.

5. Specify the adjacent surface distance.

   Loop selects all edges around the base surface and Select Edge selects individual edges.

6. Specify the edges to chamfer (2).




base surface selected | edge to chamfer selected | result

# Section and Slice 3D Solids

You can create a cross section through a 3D solid. The result can be a two-dimensional object representing the shape of the section, or it can be a 3D solid chopped in half.

With , you can create a cross section through a solid as a region or an anonymous block. The default method is specifying three points to define the plane. Other methods define the cross-sectional plane by another object, the current view, the *Z* axis, or the *XY*, *YZ*, or *ZX* plane. The cross-sectional plane is placed on the current layer.

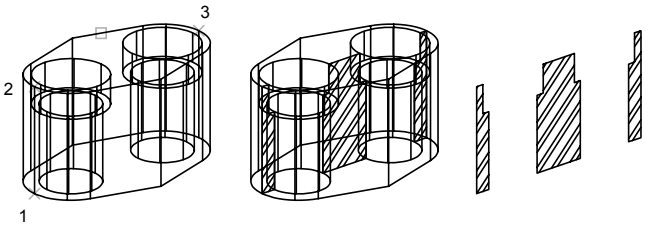
With , you can create a new solid by cutting the existing solid and removing a specified side. You can retain one or both halves of the sliced solids. The sliced solids retain the layer and color properties of the original solids. The default method of slicing a solid is to specify three points that define the cutting plane and then select which side to retain. You can also define the cutting plane by using another object, the current view, the *Z* axis, or the *XY*, *YZ*, or *ZX* plane.

**To create a cross section of a solid**

1. Click draw menu ➤ Solids ➤ Section.

2. Select the objects to cross-section.
3. Specify three points to define the cross-sectional plane.

   The first point defines the origin (0,0,0) of the cutting plane. The second point defines the *X* axis, and the third point defines the *Y* axis.




object selected and three points specified

cross-sectional cutting plane defined

cross section isolated and hatched for clarity

**NOTE** If you are applying hatching to the cross-sectional cutting plane, you must align the UCS with the cross-sectional cutting plane first.

**To slice a solid**

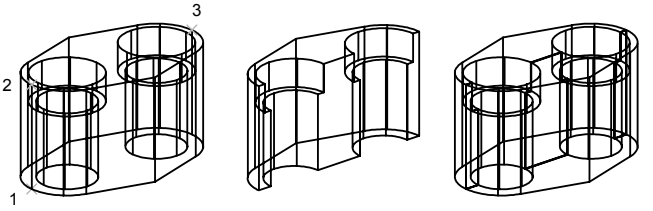
1. Click Draw menu ➤ Solids ➤ Slice.

2. Select the objects to slice.

3 Specify three points to define the cutting plane.

The first point defines the origin (0,0,0) of the cutting plane. The second point defines the positive *X* axis, and the third point defines the positive *Y* axis.

4 Specify which side to retain, or enter **b** to retain both sides.




three points specified to define the cutting plane | one half of object retained | both halves retained

# Modify Faces of 3D Solids

You can edit a 3D solid object by performing operations on selected faces of the object.

## Overview of Modifying Faces on 3D Solids

You can edit your solid object by extruding, moving, rotating, offsetting, tapering, deleting, or copying it, or by changing the color of the faces.

You can select individual faces on a 3D solid object or use one of the following selection methods:

- Boundary set
- Crossing polygon
- Crossing window
- Fence

*Boundary sets* are sets of faces defined by a closed boundary, which consists of lines, circles, arcs, elliptical arcs, and spline curves. When defining a boundary set on a solid object, you first select an internal point on the solid, highlighting the face. If you select the same point on the face again, the adjoining face is highlighted.

You can also select individual faces or edges with your pointing device or use a crossing window, an irregular shaped polygon, or a fence that selects faces or edges that it passes through.

## Extrude Faces on 3D Solids

You can extrude planar faces of a 3D solid along a path, or you can specify a height value and a tapered angle.

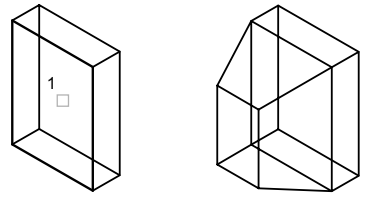
You can extrude planar faces along a path, or you can specify a height value and a tapered angle. Each face has a positive side, which is the side in the direction of the face's normal (the current face you're working on). Entering a positive value extrudes the face in its positive direction (usually outward); a negative value extrudes in the negative direction (usually inward).

Tapering the selected face with a positive angle tapers the face inward, and a negative angle tapers the face outward. The default angle, 0, extrudes the face perpendicular to its plane. If you specify a large taper angle or a long extrusion height, you can cause the face to taper to a point before it reaches the extrusion height; the extrusion is rejected. Face extrusion along a path is based on a path curve (lines, circles, arcs, ellipses, elliptical arcs, polylines, or splines).

You can also extrude the face of a solid object along the path of a specified line or curve. All profiles of the selected face extrude along the chosen path to create the extrusion. You can select lines, circles, arcs, ellipses, elliptical arcs, polylines, or splines as paths. The path should not lie on the same plane as the selected face or have areas of high curvature.

**To extrude a face on a solid object**

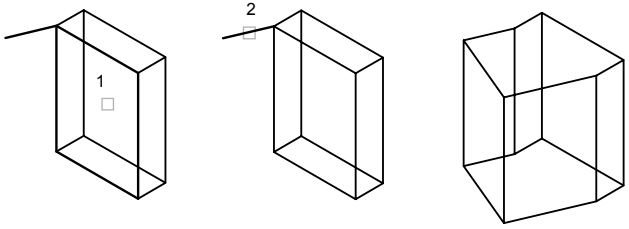
1. Click Modify menu ➤ Solids Editing ➤ Extrude Faces.
2. Select the face to extrude (1).
3. Select additional faces or press ENTER to extrude.
4. Specify the height of extrusion.
5. Specify a taper angle.
6. Press ENTER to complete the command.




face selected face extruded

**To extrude a face along a path on a solid object**

1. Click Modify menu ➤ Solids Editing ➤ Extrude Faces.
2. Select the face to extrude (1).
3. Select additional faces or press ENTER to extrude.
4. Enter **p** (Path).
5. Select the object to use as the path (2).
6. Press ENTER to complete the command.




face selected path selected face extruded

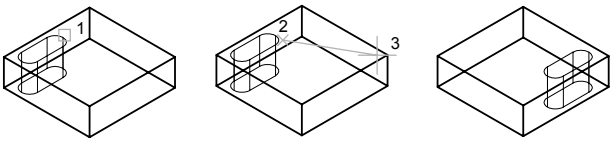
## Move Faces on 3D Solids

You can edit a 3D solid object by moving selected faces of the object.

The selected faces are moved without their orientation changing. You can easily move holes from one location to another location in a 3D solid. You can use Snap mode, coordinates, and object snaps to move the selected faces with precision.

**To move a face on a solid object**

1. Click Modify menu ➤ Solids Editing ➤ Move Faces.
2. Select the face to move (1).
3. Select additional faces or press ENTER to move the face.
4. Specify the base point for the move (2).
5. Specify the second point of displacement (3).
6. Press ENTER to complete the command.




face selected | base point and second point selected | face moved

## Rotate Faces on 3D Solids

You can rotate selected faces or a collection of features on a 3D solid object.

You rotate selected faces or a collection of features on a solid, such as holes, by choosing a base point and a relative or absolute rotation angle. All 3D faces rotate about a specified axis. The current UCS and the system variable setting determine the direction of the rotation. You can specify the axis of rotation using two points; an object; the *X*, *Y*, or *Z* axis; or the *Z* direction with respect to the line of sight in the current view.

**To rotate a face on a solid object**

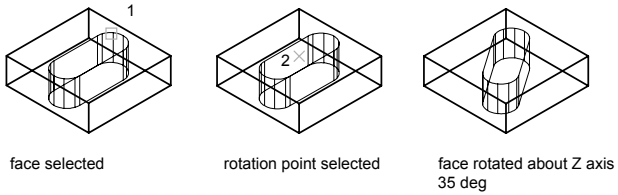
1. Click Modify menu ➤ Solids Editing ➤ Rotate Faces.
2. Select the face to rotate (1).
3. Select additional faces or press ENTER to rotate.
4. Enter **z** for the axis point.

   You can also specify the *X* or *Y* axis, two points (defines the axis of rotation), or axis by object (aligns the axis of revolution with an existing object) to define the axis point. The positive axis direction is from the

start point to the endpoint, and the rotation follows the right-hand rule, unless it is reversed by the setting of the ANGDIR system variable.

5 Specify the angle of rotation.

6 Press ENTER to complete the command.




## Offset Faces on 3D Solids

On a 3D solid, you can uniformly offset faces by a specified distance.

On a 3D solid, you can uniformly offset faces by a specified distance. New faces are created by offsetting existing ones inside or outside at a specified distance from their original positions (offsetting works in the direction of the face's normal or the positive side of the surface or face). For example, you can offset larger or smaller holes on a solid object. Specifying a positive value increases the size or volume of the solid; a negative value decreases the size or volume of the solid. You can also use a through point to specify the offset distance.

**To offset a face on a solid object**

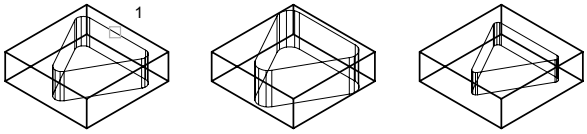
1 Click Modify menu ➤ Solids Editing ➤ Offset Faces.

2 Select the face to offset (1).

3 Select additional faces or press ENTER to offset.

4 Specify the offset distance.

5 Press ENTER to complete the command.



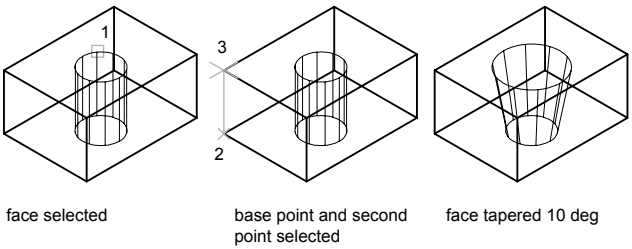

**NOTE** Holes inside a solid object offset smaller as the volume of the solid gets larger.

## Taper Faces on 3D Solids

You can taper faces with a draft angle along a vector direction. Tapering the selected face with a positive angle tapers the face inward, and a negative angle tapers the face outward. Avoid using extremely large tapered angles. If the angle is too large, the profile can taper to a point before it reaches the specified height, and the tapering is rejected.

**To taper a face on a solid object**

1. Click Modify menu ➤ Solids Editing ➤ Taper Faces.
2. Select the face to taper (1).
3. Select additional faces or press ENTER to taper.
4. Specify the base point for the taper (2).
5. Specify the second point along an axis (3).
6. Specify the angle of the taper.
7. Press ENTER to complete the command.

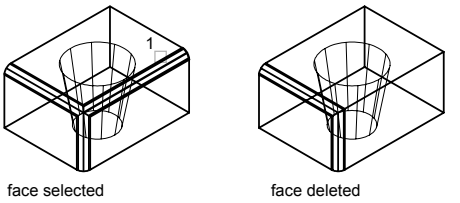


## Delete Faces on 3D Solids

You can remove faces and fillets from a 3D solid object. For example, you can use  to remove drilled holes or fillets from your 3D solid object.

**To delete a face on a solid object**

1. Click Modify menu ➤ Solids Editing ➤ Delete Faces.
2. Select the face to delete (1).
3. Select additional faces or press  ENTER  to delete.
4. Press  ENTER  to complete the command.

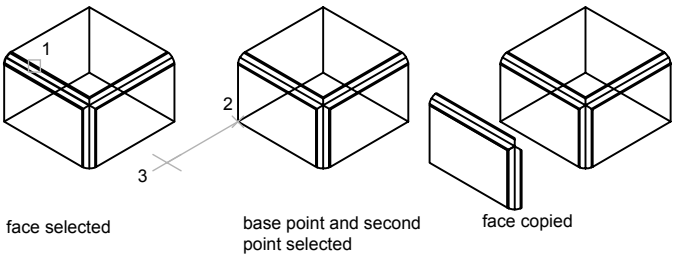



## Copy Faces on 3D Solids

You can copy faces on a 3D solid object. Selected faces are copied as regions or bodies. If you specify two points, the first point is used as a base point and a single copy is placed relative to the base point. If you specify a single point, and then press  ENTER, the original selection point is used as a base point and the next point as a point of displacement.

**To copy a face on a solid object**

1. Click Modify menu ➤ Solids Editing ➤ Copy Faces.
2. Select the face to copy (1).
3. Select additional faces or press  ENTER  to copy.
4. Specify the base point for the copy (2).
5. Specify the second point of displacement (3).
6. Press  ENTER  to complete the command.




**NOTE** Use EXTRUDE to extrude a copied face.

## Color Faces on 3D Solids

You can change the color of a selected face on a 3D solid object.

You can select a color from the seven standard colors or select a color from the Select Color dialog box. When specifying a color, you can enter the name of the color or an AutoCAD Color Index (ACI) number, an integer from 1 through 255. Setting a color on a face overrides the color setting for the layer on which the solid object resides. For information about assigning colors, see "Work with Colors" on page 287.

**To change the color of a face on a solid object**

1. Click Modify menu ➤ Solids Editing ➤ Color Faces.
2. Select the face whose color you want to change.
3. Select additional faces or press ENTER.
4. In the Select Color dialog box, select a color. Click OK.
5. Press ENTER to complete the command.

# Modify Edges of 3D Solids

You can change the color of edges or copy individual edges on a 3D solid object. Colors are selected from the Select Color dialog box. All 3D solid edges are copied as lines, arcs, circles, ellipses, or spline objects.

### Color Edges

You can assign colors to individual edges on your 3D solid object. You can select a color from the seven standard colors or select a color from the Select Color dialog box. When specifying a color, you can enter the name of the color or an ACI number, an integer from 1 through 255. Setting a color on an edge overrides the color setting for the layer on which the solid object resides. For information about assigning colors, see "Work with Colors" on page 287.

### Copy Edges

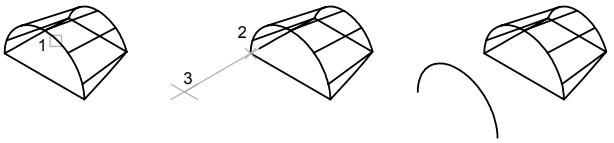
You can copy individual edges on a 3D solid object. All edges are copied as lines, arcs, circles, ellipses, or splines. If you specify two points, the first point is used as a base point and a single copy is placed relative to the base point. If you specify a single point, and then press ENTER, the original selection point is used as a base point and the next point is used as a point of displacement.

#### To change the color of an edge on a solid object

1. Click Modify menu ➤ Solids Editing ➤ Color Edges.
2. Select the edge of a face to color.
3. Select additional edges or press ENTER.
4. In the Select Color dialog box, select a color. Click OK.
5. Press ENTER to complete the command.

#### To copy an edge on a solid object

1. Click Modify menu ➤ Solids Editing ➤ Copy Edges.
2. Select the edge of the face to copy (1).
3. Select additional edges or press ENTER.
4. Specify the base point for the move (2).
5. Specify the second point of displacement (3).
6. Press ENTER to complete the command.



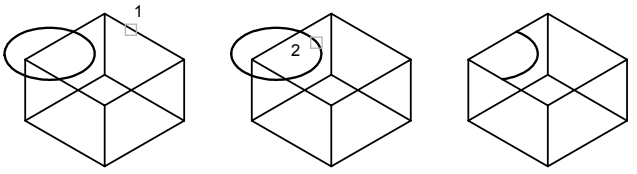

edge selected | base point and second point selected | edge copied

# Imprint 3D Solids

You can create new faces on 3D solids by imprinting arcs, circles, lines, 2D and 3D polylines, ellipses, splines, regions, bodies, and 3D solids. For example, if a circle intersects with a 3D solid, you can imprint the intersecting curves on the solid. You can delete or retain the original imprinted objects to be used for further editing. The imprinted object must intersect a face or faces on the selected solid in order for the imprinting to be successful.

**To imprint a 3D solid object**

1. Click Modify menu ➤ Solids Editing ➤ Imprint.
2. Select the 3D solid object (1).
3. Select the object that you want to imprint (2).
4. Press ENTER to retain the original objects, or enter **y** to delete them.
5. Select additional objects to imprint or press ENTER.
6. Press ENTER to complete the command.




solid selected | object selected | object imprinted on solid

# Separate 3D Solids

You can separate (take apart) composite solids. The composite 3D solid object cannot share a common area or volume. After separation of the 3D solid, the individual solids retain the layers and colors of the original. All nested 3D solid objects are separated into their simplest form.

**To separate a 3D solid composite into individual solids**

1. Click Modify menu ➤ Solids Editing ➤ Separate.
2. Select the 3D solid object.
3. Press ENTER to complete the command.

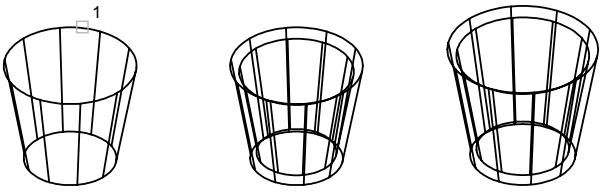
# Shell 3D Solids

You can create a shell (a hollow, thin wall with a specified thickness) from your 3D solid object. New faces are created by offsetting existing ones inside or outside their original positions. Continuously tangent faces are treated as single faces when offsetting.

**To create a 3D solid shell**

1. Click Modify menu ➤ Solids Editing ➤ Shell.
2. Select the 3D solid object.
3. Select a face to be excluded from shelling (1).
4. Select additional faces to be excluded or press ENTER.
5. Specify the shell offset valve.

   A positive offset value creates a shell in the positive face direction; a negative value creates a shell in the negative face direction.
6. Press ENTER to complete the command.



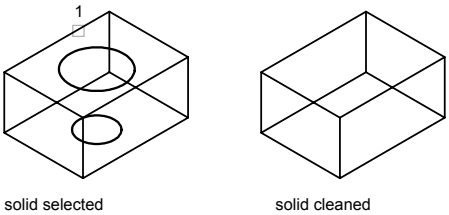

# Clean and Check 3D Solids

You can remove edges or vertices if they share the same surface or vertex definition on either side of the edge or vertex. The body, faces, or edges on the solid object are checked, and adjacent faces that share the same surface are merged. All redundant edges, imprinted as well as unused, on your 3D solid object are deleted.

You can check to see if your solid object is a valid 3D solid object. With a valid 3D solid, you can modify the object without incurring failure error messages. If the 3D solid is not valid, you cannot edit the object.

### To clean up a 3D solid object

1. Click Modify menu ➤ Solids Editing ➤ Clean.
2. Select the 3D solid object (1).
3. Press ENTER to complete the command.




### To validate a 3D solid object

1. Click Modify menu ➤ Solids Editing ➤ Check.

2 Select the 3D solid object.

3 Press ENTER to complete the command.

A message states that the solid is a valid ShapeManager solid.

# Part 6
# Hatches, Notes, Tables, and Dimensions

Chapter 21 Hatches, Fills, and Wipeouts
Chapter 22 Notes and Labels
Chapter 23 Tables
Chapter 24 Dimensions and Tolerances

# Hatches, Fills, and Wipeouts

# 22

## In this chapter

- Overview of Hatch Patterns and Fills
- Define Hatch Boundaries
- Choose Hatch Patterns and Solid Fills
- Modify Hatches and Solid-Filled Areas
- Create a Blank Area to Cover Objects

# Overview of Hatch Patterns and Fills

### Define the Boundaries of a Hatch

You can choose from several methods to specify the boundaries of a hatch.

- Specify a point in an area that is enclosed by objects.
- Select objects that enclose an area.
- Drag a hatch pattern into an enclosed area from a tool palette or DesignCenter.

When you hatch a drawing, whole or partial objects that are not part of the object boundary are ignored.

If a hatch line encounters an object such as text, an attribute, or a solid-fill object, and if the object is selected as part of the boundary set, HATCH hatches around the object.

text object not part of the boundary set

text object included in the boundary set

**NOTE** If you want to hatch an area whose boundary is not quite closed, you can set the  system variable to bridge gaps and treat the boundary as closed. HPGAPTOL applies only to gaps between lines and arcs that, if extended, would meet.

To reduce file size, a hatched area is defined in the drawing database as a single graphical object.

### Add Hatch Patterns and Solid Fills

You can use several methods to add hatch patterns to your drawing.

- The  command provides the most options.
- You can drag hatches from a tool palette. Use tool palettes when you need additional speed and convenience.

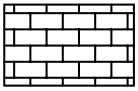
  With the Tool Palettes window open, you can right-click a pattern tool to access the Tool Properties dialog box from the shortcut menu. This dialog box contains several hatch pattern options that are also available through

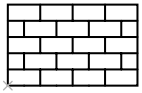
HATCH. For example, you can specify the scale and spacing for the hatch pattern.

- You can also use DesignCenter.

### Control the Hatch Origin

By default, hatch patterns always "line up" with each other. However, sometimes you might need to move the starting point, called the *origin point*, of the hatch. For example, if you create a brick pattern, you might want to start with a complete brick in the lower-left corner of the hatched area. In that case, use the Hatch Origin options in the Hatch and Gradient dialog box.


default hatch origin


new hatch origin

The location and behavior of a hatch pattern depends on the , , and system variables, and the location and orientation of the user coordinate system.

### Choose a Hatch Pattern

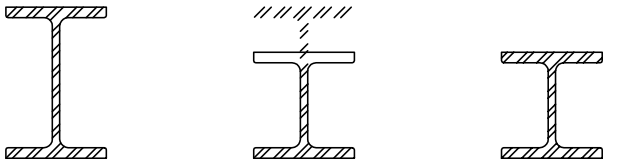
The program supplies a solid fill and more than 50 industry-standard hatch patterns that you can use to differentiate the components of objects or represent object materials. The program also comes with 14 hatch patterns that conform to the ISO (International Standards Organization) standards. When you select an ISO pattern, you can specify a pen width, which determines the lineweight in the pattern.

On the Hatch tab of the Hatch and Gradient dialog box, the Type and Pattern area displays the names of all the hatch patterns defined in the *acad.pat* text file. You can add new hatch patterns to the dialog box by adding their definitions to the *acad.pat* file.

### Create Associative Hatches

An *associative* hatch is updated when you change the boundary. Hatched areas created with HATCH are associative by default. This setting is stored in the system variable . Hatches created by dragging hatch patterns from tool palettes or DesignCenter™ use the setting in HPASSOC. You can remove hatch associativity at any time or use  to create a nonassociative hatch. When the HPGAPTOL system variable is set to 0 (the default), associativity is automatically removed if editing creates an open boundary.

You can use HATCH to create nonassociative hatches, which are independent of their boundaries.




### Assign a Draw Order to a Hatch

You can assign a draw order to a hatch so that it is drawn either behind or in front of the hatch boundary, or behind or in front of all other objects.

When you create a hatch, by default the hatch is drawn behind the hatch boundary. This makes it easier to view and select the hatch boundary. You can change the draw order of the hatch so that the hatch is drawn in front of the hatch boundary instead, or either behind or in front of all other objects. This setting is stored in the system variable. Hatches created by dragging hatch patterns from tool palettes or DesignCenter use the draw order setting in HPDRAWORDER.

### Limit Hatch Pattern Density

If you create a very dense hatch, the program may reject the hatch and display a message indicating that the hatch scale is too small or its dash length too short. You can change the maximum number of hatch lines by setting the MaxHatch system registry variable at the Command prompt by entering **(setenv "MaxHatch" "n")** where n is a number between 100 and 10000000 (ten million). The default value for MaxHatch is 10000.

---

**NOTE** When changing the value of MaxHatch, you must enter MaxHatch with the capitalization as shown.

---

### Edit Hatch Boundaries

Because there are so many combinations of objects that can be hatched, editing hatched geometry can produce unexpected results. If you create a hatch that you don't want, you can undo it, trim the hatch, or delete the hatch and rehatch the area.

### Create Custom Hatch Patterns

You can also define your own hatch pattern using the current linetype with the User Defined Pattern option of the Hatch and Gradient dialog box, or you can create more complex hatch patterns.

**See also:**

"Modify Hatches and Solid-Filled Areas" on page 693
"Overview of Hatch Pattern Definitions" in the Customization Guide

### To drag hatch patterns into your drawing

1. Click Tools menu ➤ DesignCenter.

   **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 Folders tab, click the Search button.
3. In the Search dialog box, make the following entries:
   - In Look For, select Hatch Pattern Files.
   - In the In box, select the drive where the program is installed.
   - Select the Search Subfolders option.
   - On the Hatch Pattern Files 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 on the Folders tab in DesignCenter.

5. In the search results, double-click the file to load the hatch patterns into the content area of DesignCenter.
6. (Optional) Right-click a pattern to display a shortcut menu with the following options:
   - **HATCH**. Opens the Hatch and Gradient dialog box.
   - **Copy**. Stores the hatch pattern to the Clipboard.

- **Create Tool Palette**. Creates a new tool palette with the selected pattern displayed.

7 From the content area, drag a hatch pattern onto a closed object in your drawing or onto a tool palette.

**NOTE** If the hatch pattern scale is too large or small, an error message is displayed. You can adjust the scale for any hatch pattern by double-clicking it to display the Hatch and Gradient dialog box.

Standard toolbar

**To hatch areas**

1 Click Draw menu ➤ Hatch.

2 In the Hatch and Gradient dialog box, click Add: Pick points.

3 In your drawing, specify a point inside each area that you want hatched, and then press ENTER .

This point is known as the internal point.

4 In the Hatch and Gradient dialog box, Hatch tab, in the swatch box, verify that the sample pattern is the pattern you want to use. To change patterns, select another pattern from the Pattern list.

To see how the hatch pattern will look, click the [...] button next to Pattern. Click OK when you finish previewing.

5 In the Hatch and Gradient dialog box, make adjustments, if necessary.

You can specify new hatch boundaries by clicking Add Boundaries or Remove Boundaries.

6 Under Draw Order, click one of the options.

You can change the draw order of the hatch so that the hatch is drawn either behind or in front of the hatch boundary, or behind or in front of all other objects.

7 Click OK.

Draw toolbar

**To hatch selected objects**

1. Click Draw menu ➤ Hatch.
2. In the Hatch and Gradient dialog box, click Add: Select objects.
3. Specify the object or objects you want to hatch.

   The objects need not form a closed boundary. You can also specify any islands that should remain unhatched. Also, you can set the HPGAPTOL system variable to treat a set of objects that almost enclose an area as a closed hatch boundary.
4. Under Draw Order, click one of the options.

   You can change the draw order of the hatch so that the hatch is drawn either behind or in front of the hatch boundary, or behind or in front of all other objects.
5. Click OK.

Draw toolbar

# Define Hatch Boundaries

You create a hatch by selecting an object to hatch or fill, or by defining a boundary and then specifying an internal point.

## Overview of Hatch Boundaries

You can hatch an enclosed area or hatch within a specified boundary using . By default, HATCH creates associative hatches that are updated when the boundary is changed.

You create a hatch by selecting an object to hatch or by defining a boundary and then specifying an internal point. A hatch boundary can be any combination of objects, such as lines, arcs, circles, and polylines, that forms an enclosed area.

Enclosed areas within the hatch area are referred to as islands. You can hatch them or leave them unhatched depending on the Islands setting in the Hatch and Gradient dialog box.

If you are hatching a small area in a complex drawing, you can use boundary sets to speed the process.

Objects can be hatched only if they are in a plane parallel to the *XY* plane of the current UCS.

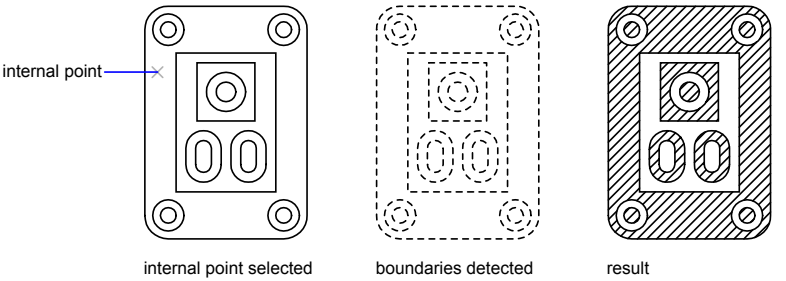
**NOTE** If you want to hatch an area that is not completely enclosed, you can set a gap tolerance ( system variable). Any gaps equal to or smaller than the value you specify in the gap tolerance are ignored, and the boundary is treated as closed.

# Control the Hatching in Islands

You can determine how *islands*, enclosed areas within the hatch boundary, are hatched using the three hatching styles: Normal, Outer, and Ignore. You can preview these hatching styles in the More Options area of the Hatch and Gradient dialog box.

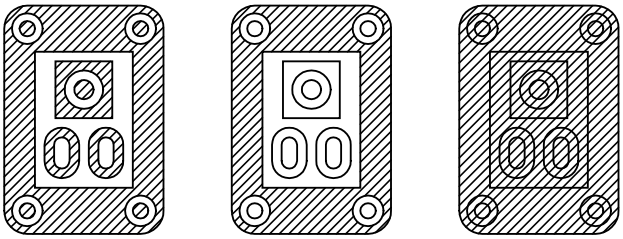
Normal hatching style (the default) hatches inward from the outer boundary. If the hatching process encounters an internal boundary, hatching is turned off until another boundary is encountered.

If you hatch using the Normal hatching style, islands remain unhatched and islands within islands are hatched, as shown below.

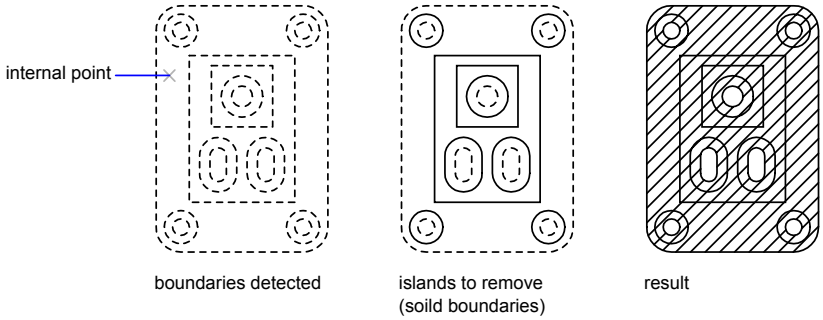



Outer hatching style hatches inward from the outer boundary and stops at the next boundary.

Ignore hatching style hatches the entire enclosed area, ignoring internal boundaries.




You can also remove any islands from the hatch area.




**See also:**

"Modify Hatches and Solid-Filled Areas" on page 693

**To remove islands from the hatch area**

1. Click Draw menu ➤ Hatch.
2. In the Hatch and Gradient dialog box, after adding one or more boundaries, click Remove Boundaries.
3. Select the boundaries that you want to remove and press ENTER.
4. In the Hatch and Gradient dialog box, click OK to apply the hatch.

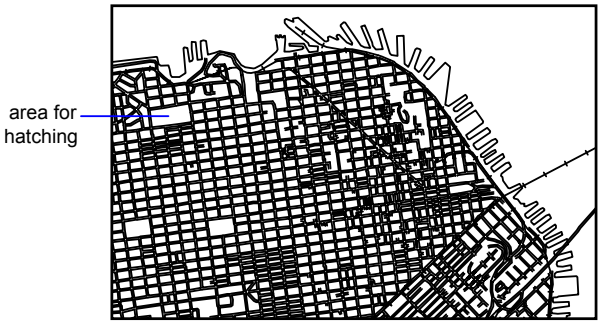
Draw toolbar

# Define Hatch Boundaries in Large Drawings

You can save time hatching a small area in a complex drawing by defining a set of objects in the drawing to be used in determining the hatch boundary.

By default, HATCH defines the boundary by analyzing all closed objects in the drawing. Analysis of all objects fully or partially visible on the screen as boundaries can be time consuming in a complex drawing. To hatch a small area of a complex drawing, you can define a set of objects in the drawing called a *boundary set*. HATCH does not analyze objects that are not included in the boundary set.

For clarity, first zoom into the area you want to hatch.




The View Selections option in the Hatch and Gradient dialog box, highlights the objects in the drawing that define the boundary.

**To define a boundary set in a complex drawing**

1. Click Draw menu ➤ Hatch.
2. In the Hatch and Gradient dialog box, More Options, under Boundary Set, click New.
3. At the Select Objects prompt, specify opposite corner points for the boundary set and press ENTER .

   If you use crossing selection by specifying points from right to left, you select all objects enclosed or crossed.

4 In the Hatch and Gradient dialog box, click Add Boundary. If necessary, enter **k** to specify the Pick Internal Point option.

5 Specify the internal point.

6 Click OK to apply the hatch.

new boundary set internal point result

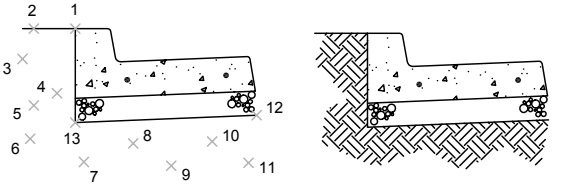
Draw toolbar

# Create Unbounded Hatches

There are several methods you can use to create a hatch that does not display a hatch boundary.

- You can create a hatch with , and then erase some or all of the boundary objects.
- You can create a hatch with , making sure that the boundary objects are on a different layer than the hatch. Then turn off or freeze the layer of the boundary objects. This is the only method that maintains hatch associativity.
- You can trim an existing hatch with objects created as trim boundaries. After trimming the hatch, erase the objects.
- You can define a hatch boundary with the Draw option of on the command line by specifying boundary points.

For example, you might want to show that a large area of a drawing is filled with a pattern by filling only a small section of that area, as shown in the following illustration.




points specified to define hatch boundary

result

You can choose whether to retain the polyline boundary after the hatch is created; here, the polyline boundary is not retained.

**To define a boundary by specifying points**

1. At the Command prompt, enter **-hatch**.
2. Enter the desired pattern. For example, enter **earth** to select the EARTH pattern.
3. Specify the scale and angle for the pattern.
4. Enter **w** to specify the Draw option.
5. Specify points to define the boundary. Enter **c** to close the polyline boundary, and then press ENTER.
6. Enter **n** to discard the polyline boundary once the hatch area has been defined, or enter **y** to create a polyline.

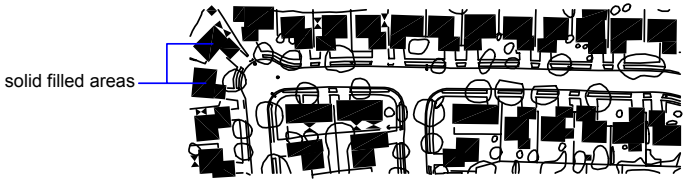
# Choose Hatch Patterns and Solid Fills

You can use a predefined hatch pattern or solid fill, or you can create your own hatch patterns.

## Create Solid-Filled Areas

Solid-filled areas can be created using

- Hatches with a solid hatch pattern ()
- 2D solids ()
- Wide polylines or donuts (, )




**See also:**

"Overview of Hatch Patterns and Fills" on page 676
"Modify Hatches and Solid-Filled Areas" on page 693
"Draw Polylines" on page 391
"Draw Donuts" on page 413

**To create a hatch with a solid pattern**

1. Click Draw menu ➤ Hatch.
2. In the Hatch and Gradient dialog box, click Add: Pick points.
3. Specify a point in your drawing inside each area that you want hatched.

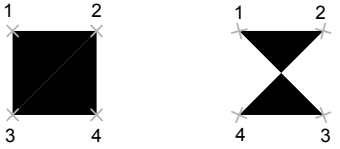
   This point is known as the internal point.
4. Press ENTER .
5. In the Hatch and Gradient dialog box, Hatch tab, under Type, click Predefined.
6. Click the [...] button next to Pattern.
7. In the Hatch Pattern Palette dialog box, Other Predefined tab, select Solid. Click OK.
8. To see how the hatch pattern will look, click Preview.
9. When you finish previewing the hatch pattern, right-click or press ENTER to apply the hatch, or press any other button or key to return to the Hatch and Gradient dialog box.
10. In the Hatch and Gradient dialog box, make adjustments, if necessary. (You can specify new hatch boundaries by clicking Add Boundaries or Remove Boundaries.)
11. Click OK.

Draw toolbar

**To create a 2D solid object**

1. At the Command prompt, enter **solid**.
2. Specify the first point.
3. Specify the second point, moving left to right.
4. Continue to specify points. Press ENTER when the object is complete.

   When you create a quadrilateral solid-filled area, the sequence of the third and fourth points determines its shape. Compare the following illustrations:




   Notice that to create the quadrilateral area, both the top and bottom edges are specified from left to right. If you specify the first point on the right and the second point on the left, then the third and fourth points should also be in a right-to-left direction. As you continue to specify pairs of points, be sure to continue this zigzag sequence to ensure the results you expect.

**To create a wide polyline**

1. Click Draw menu ➤ 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.

Draw toolbar

**To create a donut**

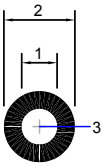
1 Click Draw menu ➤ Donut.

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.




# Create Gradient-Filled Areas

A gradient fill is a solid hatch fill that gives the blended-color effect of a surface with light on it. You can use gradient fills to suggest a solid form in two-dimensional drawings.

The color in a gradient fill makes a smooth transition from light to dark, or from dark to light, and back. You select a predefined pattern (for example, linear, spherical, or radial sweep) and specify an angle for the pattern. In a two-color gradient fill, the transition is both from light to dark and from the first color to the second.

Gradient fills are applied to objects in the same way solid fills are and can be associated with their boundaries or not. An associated fill is automatically updated when the boundary changes.

You cannot use plot styles to control the plotted color of gradient fills.

Double-click a gradient fill to modify it.

**To create a one-color gradient fill**

1. Click Draw menu ➤ Hatch.
2. In the Hatch and Gradient dialog box, click Add: Pick points or Add: Select objects.
3. Specify an internal point or select an object, and then press ENTER .
4. In the Hatch and Gradient dialog box, Gradient tab, select One Color.
5. If you want to change the color, click the [...] button next to the color.
6. In the Select Color dialog box, use the Shade/Tint slider to adjust the color.
   - Move the slider toward Tint to create a color transition that moves toward white.
   - Move the slider toward Shade to create a transition that moves toward black.
7. Click a pattern, and set the following options:
   - Select Center to create a symmetrical fill, or clear Center to move the "highlight" up and to the left.
   - Specify an angle for the "highlighted" area.
8. To see how the gradient fill will look, click Preview. Press ENTER or right-click to return to the dialog box and make adjustments.
9. When you are satisfied, in the Hatch and Gradient dialog box, click OK to create the gradient fill.

Draw toolbar

**To create a two-color gradient fill**

1. Click Draw menu ➤ Hatch.
2. In the Hatch and Gradient dialog box, click Add: Pick points or Add: Select objects.

3 Specify an internal point or select an object, and then press ENTER .

4 In the Hatch and Gradient dialog box, Gradient tab, select Two Color.

The second color is the color of the highlighted area in the gradient fill.

5 If you want to change either color, click the [...] button next to the color to open the Select Color dialog box.

6 Click a pattern, and set the following options:

- Select Center to create a symmetrical fill, or clear Center to move the "highlight" up and to the left.
- Specify an angle for the "highlighted" area.

7 To see how the gradient fill will look, click Preview. Press ENTER or right-click to return to the dialog box and make adjustments.

8 When you are satisfied, in the Hatch and Gradient dialog box, click OK to create the gradient fill.

Draw toolbar

**To modify a gradient fill**

- Double-click a gradient fill to modify it.

# Use Predefined Hatch Patterns

The program supplies a solid fill and more than 50 industry-standard hatch patterns, which can represent materials such as earth, brick, or clay.

Fourteen hatch patterns conform to International Standards Organization (ISO) standards. When you select an ISO pattern, you can specify a pen width, which determines the lineweight in the pattern.

In addition to using the patterns supplied with the program, you can use patterns from an external pattern library. These patterns are listed by name and are displayed in the Hatch Pattern Palette dialog box.

**To use a predefined hatch pattern**

1 Click Draw Menu ➤ Hatch

2 In the Hatch and Gradient dialog box, click Add: Pick points or Add: Select objects.

3 Specify an internal point or select an object.

4 In the Hatch and Gradient dialog box, Hatch tab, select Predefined in the Type box.

5 From the Pattern box, select a pattern.

6 Click OK.

Draw toolbar

# Create User-Defined Hatch Patterns

In addition to using predefined hatch patterns, you can define a simple hatch pattern based on the current linetype. You define the pattern by changing the angle and spacing of the hatch lines.

**To create a user-defined hatch pattern**

1 Specify the linetype for the user-defined hatch pattern by making that linetype current.

2 Click Draw Menu ➤ Hatch.

3 In the Hatch and Gradient dialog box, click Add: Pick points or Add: Select objects.

4 Specify an internal point or select an object.

5 In the Hatch and Gradient dialog box, Hatch tab, select User-Defined in the Type box.

6 Specify the angle and spacing of the hatch pattern.

7 To use intersecting lines in the pattern, select Double.

8 Click OK.

Draw toolbar

# Modify Hatches and Solid-Filled Areas

You can modify both the pattern fill and the boundaries of hatches. You can also modify solid-filled areas, but the method you use depends on whether the solid-filled area is a solid-filled hatch, a 2D solid, or a wide polyline or donut. You can also modify the draw order of your hatch.

### Control Hatch Pattern Density

Hatching can produce a very large number of line and point objects. Although stored as hatch objects, these line and point objects do use disk space and take time to generate. If you use a relatively small scale factor when hatching an area, the hatch could require millions of line and point objects, thus taking a very long time to complete and possibly exhausting the available resources. You can avoid this problem by imposing a limit on the number of objects created by a single  command. If the approximate number of objects needed for a particular hatch (considering the boundary extents, pattern, and scale) exceeds the limit, HATCH displays a message indicating that the hatch scale is too small or that its dash length is too short, and the hatch request is rejected. If this occurs, carefully examine your hatch settings. The scale factor may be unreasonable and may need to be adjusted.

The hatch object limit is set by the MaxHatch environment setting, which is stored in the system registry. Its default value is 10000. You can change this limit by setting the MaxHatch system registry variable using **(setenv "MaxHatch" "*n*")** where *n* is a number between 100 and 10000000 (ten million).

### Change the Hatch Properties of an Existing Hatch

You can modify hatch-specific properties, such as pattern, scale, and angle, of an existing hatch. You can use the following:

- Hatch Edit dialog box (recommended)
- Properties palette

You can also copy properties from one hatch to another. With the Inherit Properties button in the Hatch Edit dialog box, you can copy all hatch-specific properties, including the hatch origin, from one hatch to another. Use the Match Properties dialog box to copy general properties and hatch-specific properties (with the exception of the hatch origin), from one hatch to another.

You can use  to disassemble a hatch into its component objects.

### Modify a Hatch Boundary

Hatch boundaries can be copied, moved, stretched, trimmed, and so on. You can also use grips to stretch, move, rotate, scale, and mirror hatch boundaries and their associated hatches just as you do other objects. If the editing you do maintains a closed boundary, an associative hatch is updated automatically. If the editing produces an open boundary, the hatch loses any associativity with the boundary and remains unchanged. Associativity may also be lost during editing of a hatch boundary if the hatch pattern file is not available at the time of editing.

---

**NOTE** If you trim a hatch area to create a hole within it, the hole is not the same as a hatch island and the hatch loses associativity. To create an island instead, delete the existing hatch and create a new hatch with the new boundary. Also, if you trim a hatch and the hatch pattern (PAT) file is no longer available, the hatch will disappear.

---

Hatch associativity depends on whether you choose Associative in the Hatch and Gradient (HATCH) and Hatch Edit () dialog boxes. Nonassociative hatches are not updated when their original boundary is changed.

You can remove hatch associativity at any time, but once it is removed for an existing hatch, it cannot be reestablished. The hatch must be re-created to restore associativity or a new hatch boundary must be created and associated with the hatch.

To create a boundary around a nonassociative or unbounded hatch, in the Hatch and Gradient dialog box, use the Recreate Boundary option. With this option you can also specify that the new boundary is associated with the hatch.

### Modify Solid-Filled Areas

Solid-filled areas can be represented by

- Hatches (with a solid hatch pattern)
- 2D solids
- Gradient fills
- Wide polylines or donuts

You modify each of these solid-filled objects just as you would any other hatch, 2D solid, wide polyline, or donut. In addition to , you can use HATCHEDIT for solid-filled hatches and gradient fills, grip editing for 2D solids, and  for wide polylines and donuts.

### Modify the Draw Order of a Hatch

When you edit a hatch, you can change its draw order, so that it is displayed behind the hatch boundary, in front of the hatch boundary, behind all other objects, or in front of all other objects.

#### To change the angle of a hatch

1. Select the hatch pattern.
2. Right-click the hatch. Click Properties.
3. In the Properties palette, enter the new value for Angle.

Standard toolbar

#### To set the limit of objects in a hatch pattern

■ At the Command prompt, enter **(setenv "MaxHatch" "n")**

where *n* is a number between 100 and 10000000 (ten million). The higher the value, the more dense the hatch pattern.

**NOTE** MaxHatch is case-sensitive.

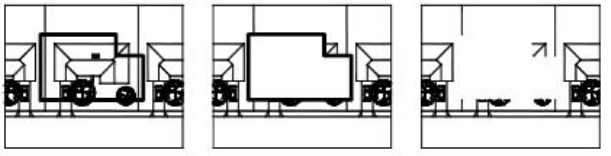
#### To modify a gradient fill

1. Double-click the gradient fill you want to modify.
2. In the Hatch Edit dialog box, Gradient tab, make any changes.
3. To see how the gradient fill will look, click Preview. Press ENTER or right-click to return to the dialog box and make adjustments.
4. When you are satisfied, in the Hatch and Gradient dialog box, click OK to create the gradient fill.

# Create a Blank Area to Cover Objects

A wipeout object is a polygonal area that masks underlying objects with the current background color. This area is bounded by the wipeout frame, which you can turn on for editing and turn off for plotting.

You can create a wipeout object by specifying a polygonal area with a series of points, or you can convert a closed polyline into a wipeout object.



Closed polyline created

Wipeout object created from polyline

Wipeout frame turned off

### Requirements and Limitations

If a polyline is used to create a wipeout object, the polyline must be closed, contain line segments only, and have zero width.

You can create wipeout objects on a layout in paper space to mask objects in model space; however, on the Plot Settings tab of the Plot dialog box, the Plot Paper Space Last option must be cleared before you plot to ensure that the wipeout object is plotted correctly.

Because a wipeout object is similar to a raster image, it has the same requirements for plotting: you need a raster-capable plotter with either an ADI 4.3 raster-capable driver or the system printer driver.

### To cover existing objects with a blank area

1. Click Draw menu ➤ Wipeout.
2. Specify points in a sequence that defines the perimeter of the area to be masked.
3. Press ENTER to end.

### To turn all wipeout frames on or off

1. Click Draw menu ➤ Wipeout.
2. On the command line, enter **f** (Frames).
3. Enter **on** or **off** and press ENTER .

# Notes and Labels

# 23

You can create and modify several types of text, including text with leaders. You can control most text style settings by defining text styles.

## In this chapter


- Overview of Notes and Labels
- Create Text
- Use Fields in Text
- Work with Text Styles
- Change Text
- Check Spelling
- Use an Alternate Text Editor

# 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 (also called *mtext)*. You can also create multiline text with leaders.

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.

# Create Text

You can create text using several methods, depending on your needs.

**See also:**

"Use Fields in Text" on page 725

# Overview of Creating Text and Leaders

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.

### Leader Objects

A leader object is a line or a spline with an arrowhead at one end and a multiline text object at the other. The leader object is associated with the multiline text object, so when the text object is moved, rotated, or scaled, the leader object is updated accordingly. Similarly, when associative dimensioning is turned on, and object snaps are used to locate the leader arrowhead, a leader object is also associated with any objects to which the arrowhead is attached.

You can copy text used elsewhere in a drawing and append a leader to it.

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**NOTE** The leader object should not be confused with the leader line that is automatically generated as part of a dimension line.

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# Create Single-Line Text

Use single-line 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** on the command line to enter the text on the command line.

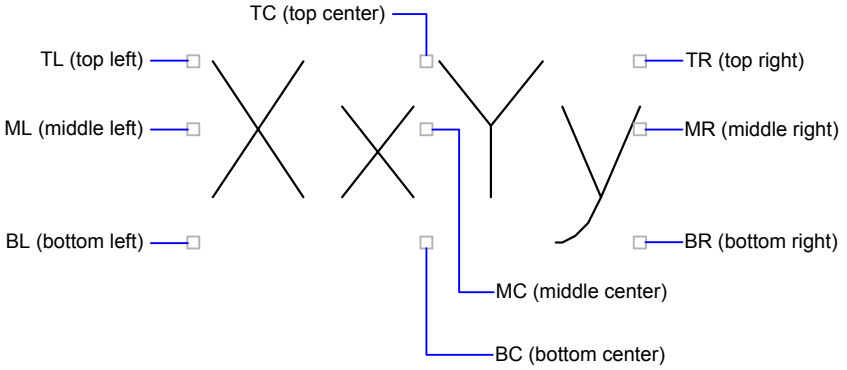
You can insert a field in single-line text. A field is text that is set up to display data that might change. When the field is updated, the latest value of the field is displayed.

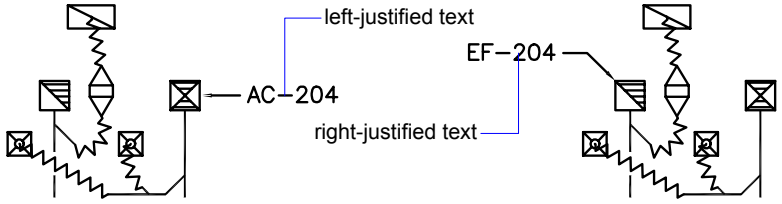
The text styles used for single-line text are the same as those used for multiline text. When you create text, you assign an existing style by entering its name at the Style prompt. If you need to apply formatting to individual words and characters, use multiline text instead of single-line text.

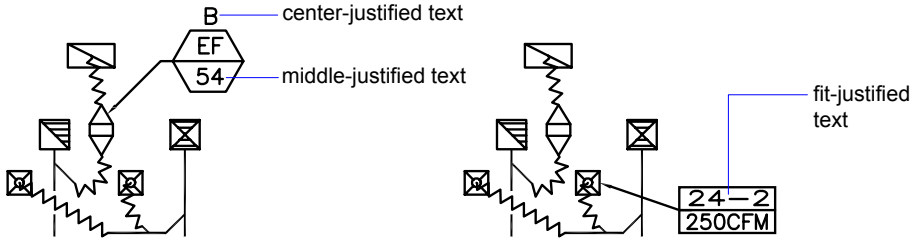
You can also compress single-line text to fit between points that you specify. This option stretches or squeezes the text to fill the designated space.

### 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 725

**To create single-line text**

1 Click Draw menu ➤ Text ➤ 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.

   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 Draw menu ➤ Text ➤ 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 Draw menu ➤ Text ➤ 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.

## 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 one or more paragraphs of multiline text (mtext) in the In-Place Text Editor (or an alternative text editor) or use prompts on the command line. You can also 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.

The In-Place Text Editor displays the bounding box with a ruler at the top and the Text Formatting toolbar. 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, check 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 set tabs and indent text to control the appearance of paragraphs in the multiline text object.

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.

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**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 Draw menu ➤ Text ➤ 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. 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, click the arrow next to the Text Style control on the toolbar and then select a style.
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 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 toolbar. SHX fonts do *not* support boldface or italics.
   - To apply color to selected text, choose a color from the Color list. Click Other to display the Select Color dialog box.

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 .

Draw toolbar

**To insert symbols or special characters in multiline text**

1 Double-click a multiline text object to open the In-Place Text Editor.

2 On the 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:

- Click OK on the toolbar.
- Click in the drawing outside the editor.
- Press CTRL + ENTER .

Draw toolbar

**To add an opaque background or fill to a multiline text object**

1. Double-click a multiline text object to open the In-Place Text Editor.
2. 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:

   - Click OK on the toolbar.
   - Click in the drawing outside the editor.
   - Press CTRL + ENTER .

   The opaque background is applied when you exit the editor.

Draw toolbar

## Justify Multiline Text

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.

top left: left-justified, spills down

top center: center-justified, spills down

top right: right-justified, spills down

middle left: left-justified, spills up and down

middle center: center-justified, spills up and down

middle right: right-justified, spills up and down

bottom left: left-justified, spills up

bottom center: center-justified, spills up

bottom right: right-justified, spills up

**To justify multiline text**

1. Click Tools menu ➤ Properties.
2. Select the multiline text object.
3. On the Properties palette, select one of the Justification options.
4. Click outside the Properties palette.

## Format Characters Within Multiline Text

As you create multiline text, you can override the text style and apply different formatting to individual words and characters. 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 .

**See also:**

"Use an Alternate Text Editor" on page 751

### To format characters in multiline text

1. Double-click a multiline text object to open the In-Place Text Editor.
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 the toolbar, make format changes as follows:
   - To change the font of the selected text, select a font from the list.
   - To change the height of the selected text, enter a new value in the Height box.
   - To format text in a TrueType font with boldface or italics, or to create underlined or overlined text for any font, click the corresponding button on the toolbar. SHX fonts do *not* support boldface or italics.
   - To apply color to selected text, 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:

- Click OK on the toolbar.
- Click in the drawing outside the editor.
- Press CTRL + ENTER .

Draw toolbar

## Create Lists 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 also remove and reapply list formatting.

By default, list formatting is applied to all text that looks like a list. Text that meets the following criteria is considered to be a list. The line begins with (1) one or more letters or numbers or a symbol, followed by (2) punctuation after a letter or number, (3) a space created by pressing TAB, and (4) some text before the line is ended by ENTER or SHIFT+ENTER.

---

**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 formatted lists in the multiline text object, and existing lists are converted to plain text. The plain text lists retain their bullets, numbers, or letters, but if you add or delete list items, the numbering does not adjust automatically.

---

List formatting is only available when the Allow Bullets and Lists option is checked (the default). 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, you can type **421.**, press TAB, and enter some text. When you press ENTER, the next line starts with 422, a period, and a TAB space.

The following characters can be used as punctuation after the number or letter when you type a list but cannot be used as bullets:

| Character | Description |
|---|---|
| . | Period |
| : | Colon |
| ) | Close parenthesis |
| > | Close angle bracket |
| ] | Close square bracket |
| } | Close curly bracket |

### Paste a List from Another Document

If you copy a nested bulleted list (a list within a list) from Microsoft® Word and paste the list into multiline text, the bullets that are displayed as empty circles cannot be formatted like other bullets in multiline text. This is because Word uses the letter *o* instead of a bullet for nested bulleted lists. You can remove formatting from the nested list and reapply to change the bullets to double bullets.

### To format multiline text as a list

1. Click Draw menu ➤ Text ➤ 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:

- Click OK on the toolbar.
- Click in the drawing outside the editor.
- Press CTRL + ENTER .

Draw toolbar

**To remove list formatting from multiline text**

1 Double-click a multiline text object to open the In-Place Text Editor.

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:

- Click OK on the toolbar.
- Click in the drawing outside the editor.
- Press CTRL + ENTER .

Draw toolbar

**To create a lettered or numbered list in multiline text as you type**

1 Click Draw menu ➤ Text ➤ 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 OK on the toolbar.
- Click in the drawing outside the editor.
- Press CTRL + ENTER .

Draw toolbar

**To create a bulleted list in multiline text as you type**

1. Click Draw menu ➤ Text ➤ 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. Start a line of text by entering **\U+2022** (the Unicode string for a bullet) or by selecting a bullet character or another symbol.

   The following characters cannot be used as bullets: 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 bullet character is automatically added to the next item.
8. Press ENTER twice to end the list.
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 .

Draw toolbar

**To move a list item in multiline text down a level**

1. Double-click a multiline text object to open the In-Place Text Editor.
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:
   - Click OK on the toolbar.
   - Click in the drawing outside the editor.
   - Press CTRL + ENTER .

Draw toolbar

**To separate an existing list**

1. Double-click a multiline text object to open the In-Place Text Editor.
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 .

Draw toolbar

**To convert the lists in a multiline text object to plain text**

1. Double-click a multiline text object to open the In-Place Text Editor.
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.

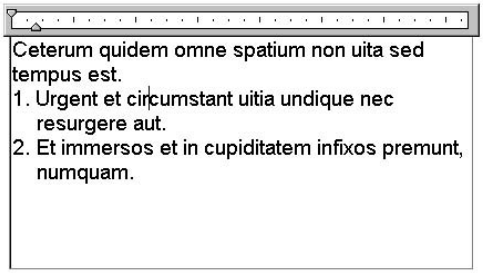
## Indent Multiline Text and Use Tabs

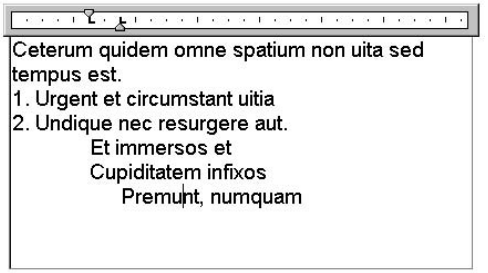
You can control how paragraphs are indented in a multiline text (mtext) object. The ruler in the In-Place Text Editor shows the settings for the current paragraph.

Tabs and indents that you set before you start to enter text apply to the whole multiline text object. To apply different tabs and indents to individual paragraphs, click in a single paragraph or select multiple paragraphs and then change the settings.

Sliders on the ruler show indentation relative to the left side of the bounding box. The top slider indents the first line of the paragraph, and the bottom slider indents the other lines of the paragraph.

The long tick marks on the ruler show the default tab stops. If you click the ruler to set your own tabs, the ruler displays a small, L-shaped marker at each custom tab stop. You can delete a custom tab stop by dragging the marker off the ruler.



**To create paragraphs with hanging indentation**

1. Double-click a multiline text object to open the In-Place Text Editor.
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:

   - Click OK on the toolbar.
   - Click in the drawing outside the editor.
   - Press CTRL + ENTER .

Draw toolbar

## 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 Tools menu ➤ Properties.

   The Properties palette is displayed.

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.

---

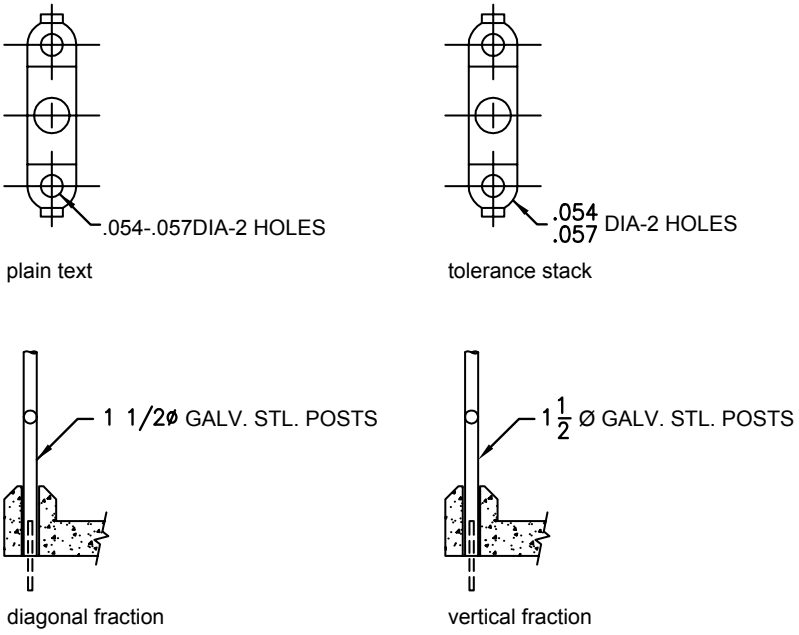
Standard toolbar

# Create Stacked Characters Within Multiline Text

Characters representing a fractional measurement or tolerance can be formatted to conform to several standards.

Stacked text is text or fractions that indicate tolerances or measurements. You use special characters to indicate where 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 not separated by a line.




You can set the program to automatically stack numeric characters entered before and after the slash, pound sign, or carat. For example, if you enter **1#3** followed by a nonnumeric character or space, the AutoStack Properties dialog box is displayed. You can change the settings to indicate your formatting preferences.

AutoStack stacks only numeric characters immediately before and after the slash, pound sign, and carat. To stack nonnumeric characters or text that includes spaces, select the text and click the Stack button on the Text Formatting toolbar.

**See also:**

"Use an Alternate Text Editor" on page 751

**To create stacked text**

1. Click Draw menu ➤ Text ➤ Multiline Text.
2. Specify opposite corners of a bounding box to define the width of the multiline text object.
3. In the In-Place Text Editor, set text style and other multiline text properties as needed.
4. Enter the text you want to stack separated by one of the following characters:

   - Slash (/) stacks text vertically, separated by a horizontal line.
   - Pound sign (#) stacks text diagonally, separated by a diagonal line.
   - Carat (^) creates a tolerance stack, which is not separated by a line.

   If you enter numbers separated by stack characters and then enter a nonnumeric character or press SPACEBAR, the AutoStack Properties dialog box is displayed.
5. In the AutoStack Properties dialog box, you can choose to automatically stack numbers (not nonnumeric text) and to remove leading blanks. You can also specify whether the slash character creates a diagonal fraction or creates a vertical fraction. If you do not want to use AutoStack, click Cancel to exit the dialog box.
6. Select the text that you want to stack, and click the Stack button on the toolbar.
7. To save your changes and exit the editor, use one of the following methods:

   - Click OK on the toolbar.
   - Click in the drawing outside the editor.
   - Press CTRL + ENTER .

Draw toolbar

**To change stack properties**

1. Double-click the multiline text object you want to edit.
2. In the 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 .

Draw toolbar

**To unstack text**

1. Double-click the multiline text object you want to edit.
2. In the 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 .

Draw toolbar

# Create Text with Leaders

You can create text that includes one or more leader lines.

## Overview of Text with Leaders

A leader object is a line or a spline with an arrowhead at one end and a multiline text object at the other. In some cases, a short horizontal line, called a hook line, dogleg, or landing, connects text and feature control frames to the leader line.

The leader line is associated with the multiline text object, so when the text object is relocated, the leader line stretches accordingly.

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 leader line stretches accordingly.

You can copy text used elsewhere in a drawing and append a leader to it.

**See also:**

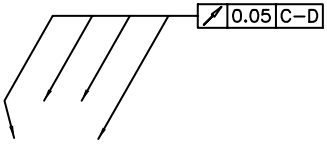
"Change Text with a Leader" on page 747

## Create a Leader Line

A leader object typically consists of an arrowhead, a leader line or curve, and a multiline text object.

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. Leader color is controlled by the current dimension line color. Leader scale is controlled by the overall dimension scale set in the current dimension style. The type and size of arrowhead, if one is present, are controlled by the first arrowhead defined in the current dimension style.

LEADER creates complex leader lines that can be made up of more than two line segments. and create simple automatic leader lines with two line segments for circles and arcs.




### Associate Leaders with Objects

When associative dimensioning is turned on (), 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.

### To create a leader with straight lines

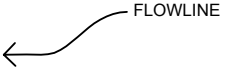
1. Click Dimension menu ➤ Leader.
2. Press ENTER to display the Leader Settings dialog box and make the following selections:
   - On the Leader Line & Arrow tab, select Straight. Under Number of Points, select No Limit.
   - On the Annotation tab, select MText.
   - Click OK.
3. Specify the First leader point and Next points of the leader line.
4. Press ENTER to finish selecting leader points.
5. Specify the text width.
6. Enter the line of text. Press ENTER to enter new lines of text as needed.
7. Press ENTER twice to end the command.

   After the QLEADER command is completed, the text annotation becomes a multiline text object.

Dimension toolbar

**To create a two-segment spline leader with text**

1. Click Dimension menu ➤ Leader.
2. Press ENTER to display the Leader Settings dialog box, and make the following selections:
   - On the Annotation tab, select MText.
   - On the Leader Line & Arrow tab, select Spline. Under Number of Points, in the Maximum box, enter **3**. Clear the No Limit option if necessary.
3. Click OK.
4. Specify the first, second, and third (optional) leader line points.
5. Specify the width of the text column.
6. Enter the first line of text. To add additional lines, press ENTER once.
7. Press ENTER twice to end the command.




Dimension toolbar

**To create multiple leaders from the same annotation**

1. Select the leader and then the leader arrowhead grip.
2. At the Command prompt, enter **c** to select the Copy option.
3. Specify the endpoints for the multiple leaders, and then press ENTER.
4. To move the end of the new leader to the hook line (dogleg or landing), clear any grips by pressing ESC, select the new leader line, select the leader line grip, and move the grip to the hook line.

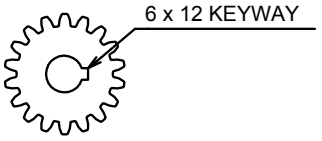
Dimension toolbar

## Add Text to a Leader

There are two ways to create leader text. You can enter text on the command line, or you can create paragraphs of text using the In-Place Text Editor. If you create multiline text, you can apply formatting to individual words or letters. You can also append copies of existing annotation to a leader.

Text is placed automatically at the endpoint of the leader, at the specified offset. You specify this offset in the Modify Dimension Style Manager, Text tab, under Text Placement. Feature control frames also are placed automatically at the endpoint of the leader. Blocks are inserted at the location, scale, and rotation you specify.

Use the Vertical settings in the Modify Dimension Style Manager, Text tab, under Text Placement to align the annotation vertically with the leader. In the following illustration, the vertical justification option used is Above.




To create a box around the text, set the  system variable to a negative value. The offset between the box and the text is the absolute value of DIMGAP.

**To create a leader using the In-Place Text Editor**

1. Click Dimension menu ➤ Leader.
2. Specify the First leader point and Next points of the leader.
3. Press ENTER to end the leader.
4. Specify the text width.
5. At the Enter First Line of Annotation prompt, press ENTER.
6. In the In-Place Text Editor, enter and format the text.
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 .

The text is appended to the leader endpoint at the offset distance specified in the Dimension Style Manager (DIMSTYLE).

Dimension toolbar

# Import Text from External Files

You can insert TXT or RTF text files created in word processors into your drawing by either importing the text or dragging a file icon from Windows Explorer.

Importing TXT or RTF files from other sources gives you the most flexibility. For example, you can create a text file of standard notes that you include in drawings. The imported text becomes a multiline text object, which you can edit and reformat. Text imported from a TXT file inherits the current text style. Text imported from an RTF file inherits the current text style *name*, but retains its original fonts and format. Imported text files are limited to 32 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 1000

**To import text files**

1. Click Draw menu ➤ Text ➤ 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 32 KB.
4. In the Select File dialog box, double-click the file you want to import, or select the file. Click Open.

The text is inserted at the cursor location.

5 Change the text as needed.

6 To save your changes and exit the editor, use one of the following methods:

- Click OK on the toolbar.
- Click in the drawing outside the editor.
- Press CTRL + ENTER .

Draw 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.

# Use Fields in Text

A field is updatable text that is set up to display data that may change during the life cycle of the drawing. When the field is updated, the latest value of the field is displayed.

## Insert Fields

A field is text that contains instructions to display data that you expect to change during the life cycle of the drawing.

When a field is updated, the latest data is displayed. For example, the value of the FileName field is the name of the file. If the file name changes, the new file name is displayed when the field is updated.

Fields can be inserted in any kind of text (except tolerances), including text in table cells, attributes, and attribute definitions. When any text command is active, Insert Field is available on the shortcut menu.

Some sheet set fields can be inserted as placeholders. For example, you can insert SheetNumberAndTitle as a placeholder. Later, when the layout is added to a sheet set, the placeholder field displays the correct sheet number and title.

Block placeholder fields can be used in block attribute definitions while you're working in the Block Editor.

A field for which no value is available displays hyphens (----). For example, the Author field, which is set in the Drawing Properties dialog box, may be blank. An invalid field displays pound signs (####). For example, the CurrentSheetName field, which is valid only in paper space, displays pound signs if it is placed in model space.

### Change the Appearance of a Field

The field text uses the same text style as the text object in which it is inserted. By default, fields are displayed with a light gray background that is not plotted ( 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

Because a field is part of a text object, it cannot be selected directly. The text object must be selected and an editing command must be active. When a field is selected, Edit Field is available on the shortcut menu, or you can double-click the field to display the Field dialog box. Any changes are applied to all text in the field.

If you no longer want to update a field, you can preserve the value that is currently displayed by converting the field to text.

The field expression, consisting of escape characters and a field code, is shown in the Field dialog box but cannot be edited.

#### To insert a field in text

1. Double-click the text to display the appropriate text editing dialog box.
2. Place the cursor where you want the field text to appear and right-click. Click Insert Field.

   For keyboard access, press CTRL + F .
3. In the Field dialog box, in Field Category, select All or select a category.

The fields in the selected category are displayed in the Field Names list.

4 In the Field Names list, select a field.

The current value of most fields is displayed in a shaded text box to the right of Field Category. The current value of a date field is displayed in the Examples list.

5 Select a format and any other options.

For example, when the NamedObject field is selected, you select a type (for example, layer or textstyle) and a name (for example, 0 for layer or STANDARD for textstyle).

Field Expression displays the expression that underlies the field. The field expression cannot be edited, but you can learn how fields are constructed by viewing this area.

6 Click OK to insert the field.

The field displays its current value in the text when the Field dialog box closes.

### To insert a field in a table

1 Double-click inside a cell in a table to select it for editing.

2 Place the cursor where you want the field text to appear and right-click. Click Insert Field.

3 In the Field dialog box, select All or select a category.

4 In the Field Names list, select a field.

The current value of the field is displayed in a shaded text box to the right of Field Category.

5 Select a format and any other option.

6 Click OK to insert the field.

The field displays its current value when you move to the next cell.

### To use a field to display a property of an object

1 Double-click a text object to display the appropriate text editing dialog box.

2 Place the cursor where you want the field text to appear and right-click. Click Insert Field.

3 In the Field dialog box, in Field Category, select All.

4 In the Field Names list, select Object.

5 In Object Type, click the Select Object button, and select an object in the drawing.

6 In the Field dialog box, in Property, select the property whose value you want to display in the text.

For example, the field could display the radius of a selected circle.

7 Select a format for the text.

8 Click OK.

The current value for the object's property is displayed in the text.

### To insert a sheet set placeholder field

1 On the Draw menu, click Block ➤ Define Attributes.

2 In the Attribute Definition dialog box, under Mode, click Preset. Specify any text options.

3 Under Attribute, in the Tag box, enter a name for the field.

4 To the right of the Value box, click the Insert Field button.

5 In the Field dialog box, under Field Category, select SheetSet.

6 In the Field Names list, select SheetSetPlaceholder.

7 In Placeholder Type, select the type of field that you want.

Temporary Value previews the placeholder field.

8 In Format, select a capitalization style for the placeholder field.

When the field is updated, the same style is used for the value of the field.

9 (Optional) Click Associate Hyperlink if you want the field to serve as a hyperlink to a location.

**NOTE** Associate Hyperlink is not available for the ViewportScale field.

10 Click OK to exit the Field dialog box.

The placeholder field displays its name when the Field dialog box closes: for example, SheetNumber. Later, when the block is inserted from the Views List tab shortcut menu in the Sheet Set Manager, the field displays a value based on the sheet it is dragged to: for example, its sheet number.

11 Click Ok in the Attribute Definition dialog box. Specify the location for the field text to appear near the geometry that you have created to use with the Sheet Set Manager as a callout block or a label block.

12 Create a block that includes the newly created attribute object along with the geometry for the block.

### To format a field value

1 Double-click a text object to display the appropriate text editing dialog box.

2 Double-click the field you want to format.

The Field dialog box is displayed. If formatting is available for the field, the Field Format button is displayed.

3 Click Field Format.

In the Field 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.

# Update Fields

You can also set fields to be updated automatically when the drawing is opened, saved, plotted, regenerated, or sent via .

Settings on the User Preferences Tab (Options Dialog Box) control whether fields are updated automatically or on demand ( system variable). The Date field cannot be updated automatically regardless of the setting of FIELDEVAL.

**NOTE** When the system variable is set to 2, fields cannot be updated until you use or .

### Contextual Fields in Blocks and Xrefs

Some fields are contextual; that is, their value is different depending on which space or layout they reside in. For example, because each layout can have a different page setup attached, the value displayed by the PlotOrientation field can be different in different layouts in the same drawing.

| List of contextual fields | |
|---|---|
| CurrentSheetCustom | CurrentSheetTitle |
| CurrentSheetDescription | DeviceName |
| CurrentSheetNumber | PaperSize |
| CurrentSheetNumberAndTitle | PlotDate |
| CurrentSheetSet | PlotOrientation |
| CurrentSheetSetCustom | PlotScale |
| CurrentSheetSetDescription | PageSetupName |
| CurrentSheetSubSet | PlotStyleTable |

For compatibility with previous versions, contextual fields in blocks and xrefs are not updated when you insert them in a drawing; instead, the field displays the last cached value. Therefore, if you want to use a contextual field within a block, for example, a title block, you must insert the field as an attribute. For example, a title block can use the CurrentSheetNumber field as an attribute. When you insert the title block, the field displays the sheet number of the sheet on which the title block is inserted.

Most fields are not contextual and are updated in blocks and xrefs. Fields in xrefs are updated based on the host file, not the source xref. These fields do not have to be placed in attributes. For example, a field that displays the sheet number of a particular sheet in a sheet set and that updates if that sheet

number changes, is a property of the sheet set. When you create the field, you select the SheetSet field name, select the sheet set and the sheet that you want in the tree view, and then select the property SheetNumber for the field value to be displayed. This field displays the sheet number of that sheet, even if you put the field in a block and insert it in another drawing. If the sheet is removed from the sheet set, it no longer has a sheet number, and the field becomes invalid and displays pound signs.

Some sheet set fields can be inserted as placeholders. For example, when you create your own callout blocks and label blocks, you can insert the SheetNumber field as a placeholder. Later, when the block is inserted from the Views List tab shortcut menu in the Sheet Set Manager, the field displays the sheet number of the drawing.

#### Compatibility with AutoCAD LT and with Previous Releases of AutoCAD

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 AutoCAD LT. When a drawing with fields is opened in AutoCAD LT, the fields are evaluated based on the setting of the FIELDEVAL system variable in the drawing, but the FIELDEVAL system variable is not accessible.

**See also:**

"Work with AutoCAD Drawings in AutoCAD LT"

#### To update a field manually

1. Double-click text.
2. Select the field to update and right-click. Click Update Field.

#### To update multiple fields manually

1. On the command line, enter **updatefield**.
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. On the command line, 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**.

# Use Hyperlinks in Fields

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 command can create a hyperlink with a relative path.

The fields that point to sheet and view titles and numbers can be assigned a hyperlink when they are created. When these items are changed or moved in the Sheet Set Manager, the hyperlinks associated with them still jump to the correct location.

Hyperlinks in fields are converted from links to DWG files into links to DWF files when published to a multi-sheet DWF file format.

### To add a hyperlink field to text

1 Click Draw menu ➤ Text ➤ 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.

Draw toolbar

# 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, which 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 prompts on the command line. 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  or by deleting the text styles from the Text Styles dialog box. The STANDARD text style cannot be removed.

### Change Text Style

When you change the text style of a multiline text object, the updated settings are applied to the entire object, and some formatting of individual characters might not be retained. The following table describes the effects of text style change on character formatting.

| Formatting | Retained? |
|---|---|
| Bold | No |
| Color | Yes |
| Font | No |
| Height | No |
| Italic | No |
| Stacking | Yes |
| Underlining | Yes |

**To set the current text style**

- On the Styles toolbar, in the Text Style control, click the arrow and select a text style from the list.

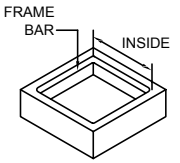
# Assign Text Fonts

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.

## 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  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 740

**To assign a TrueType font to a text style**

1. Click Format menu ➤ 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 a TrueType font from the list. TrueType fonts display a TrueType icon in front of their names.
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.

Styles toolbar

## Use Unicode and Big Fonts

Several factors affect your choosing, entering, and displaying Unicode and big fonts in a drawing.

The program supports the Unicode character-encoding standard. A Unicode font can contain 65,535 characters, with shapes for many languages. Unicode fonts 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. All SHX shape fonts are now Unicode fonts.

The text files for some alphabets, such as Kanji, contain thousands of non-ASCII characters. To accommodate such text, the program supports 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.

When you specify fonts using STYLE Command Line, 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 on the command line | |
|---|---|
| **Enter this ...** | **To specify this ...** |
| [font name],[big font name] | Both normal fonts and Big Fonts |
| [font name], | Only a normal font (Big Font unchanged) |
| ,[big font name] | Only a Big Font (normal font unchanged) |
| [font name] | Only a normal font (Big Font, if any, removed) |
| ENTER (null response) | No change |

**NOTE** Long file names that contain commas as font file names are not accepted. The comma is interpreted as a separator for an SHX font–Big Font pair.

**See also:**

"Substitute Fonts" on page 738

### To assign an Asian-language font to a text style

1. Click Format menu ➤ 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 a 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.

Styles toolbar

# Substitute Fonts

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  system variable. If you use a text style that uses a Big Font, you can map it to another font using the FONTALT system variable. This system variable uses a default font file pair of *txt.shx*, *bigfont.shx*. For more information, see "Use Unicode and Big Fonts" on page 736.

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 that comes with the program.

### 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 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.

### Use Font Mapping Files

Sometimes you might want to ensure that your drawing uses only certain fonts, or perhaps you might want to convert the fonts you used to other fonts. You can use any text editor to create font mapping tables for both of these purposes.

You can use these font mapping files to enforce corporate font standards, or to facilitate off-line printing. For example, if you share drawings with consultants, you may want to use a font mapping table to specify what font is substituted when a text object created with another font is encountered. Similarly, you can edit the drawing using quicker-drawing SHX fonts and then switch to more complex fonts for the final plot by setting up a font mapping file that converts each SHX font to an equivalent.

The font mapping file is a plain ASCII text (FMP) file containing one font mapping per line. Each line contains the name of a font file (with no directory name 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*.

For example, you could use the following entry in a font map table to specify that the *times.ttf* TrueType font file be substituted for the *romanc.shx* font file:

romanc.shx;times.ttf

The program comes with a default font mapping file. You can edit this file using any ASCII text editor. You can also specify a different font mapping file in the Options dialog box by using the  system variable. The following table shows the font substitution rules used if a font file cannot be located when a drawing is opened.

| Font substitution | | | | |
|---|---|---|---|---|
| **File extension** | **First mapping order** | **Second mapping order** | **Third mapping order** | **Fourth mapping order** |
| *.ttf* | Use font mapping table | Use font defined in text style | Windows substitutes a similar font | |
| *.shx* | Use font mapping table | Use font defined in text style | Use FONTALT | Prompt for new font |
| *.pfb* | Use font mapping table | Use FONTALT | Prompt for new font | |

**To specify a font mapping file**

1. Click Tools menu ➤ Options.
2. On the Files tab, in the list, double-click Text Editor, Dictionary, and Font File Names.
3. Double-click Font Mapping File.

   The *acad.fmp* file is specified by default.
4. To change the font mapping file, double-click the arrow line to open the Select a File dialog box. Select a file and click Open. Click OK.
5. At the Command prompt, enter **regen** to convert existing text using the new font mappings.

**To specify a default alternate font**

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

2 Enter the name of the font file you want to use as the alternative.

# Set Text Height

Text height determines the size in drawing units of the letters in the font you are using. Except in 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 Format menu ➤ 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.

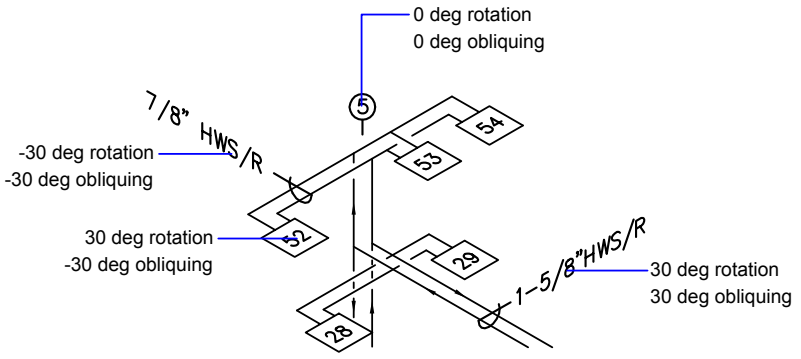
4 To update existing text that uses this text style, click Apply.

5 Click Close.

Styles toolbar

# 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**

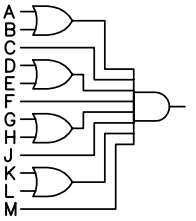
1. Click Format menu ➤ Text Style.
2. In the Text Style dialog box, select a text style from the Style Name list.
3. Under Effects, enter an angle between -85 and 85 in the Oblique Angle box.

   A positive value slants text to the right. A negative value slants text to the left.
4. To update existing text that uses this text style, click Apply.
5. Click Close.

Styles toolbar

## Set Horizontal or Vertical Text Orientation

Lines of text are oriented to be vertical or horizontal. Text can have a vertical orientation only if the associated font supports dual orientation. You can create more than one line of vertical text. Each successive text line is drawn to the right of the preceding line. The normal rotation angle for vertical text is 270 degrees.



vertical text

**NOTE** Vertical orientation is not supported for TrueType fonts and symbols.

**To set vertical orientation in a text style**

1. Click Format menu ➤ Text Style.
2. In the Text Style dialog box, select a text style from the Style Name list.
3. Under Effects, select Vertical.
4. To update existing text that uses this text style, click Apply.
5. Click Close.

Styles toolbar

# Change Text

You can change text content, formatting, and properties such as scale and justification.

# Overview of Changing Text

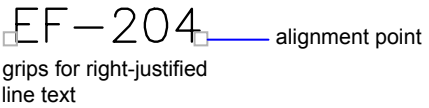
Text, whether created with , , , or , can be modified like any other object. You can move, rotate, erase, and copy it. You can change text properties in the Properties palette.

You can also edit the contents of existing text and create a mirror image of it. The  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.

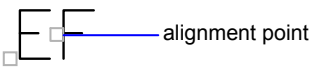
# Change Single-Line Text

You can change single-line text with  and . 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.




grips for right-justified line text




grips for middle-justified line text

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.

Standard toolbar

# 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. You can easily find and replace text with the FIND command.

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 the In-Place Text Editor 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  or  create multiline text automatically without requiring that a bounding box be specified; these objects have only a single grip at the justification point.

### Find and Replace Text

To search for and replace text, use . Replacement is based on text content only; character formatting and text properties are not changed.

With FIND, you can use wild-card characters in your search.

| Character | Definition |
|---|---|
| # (Pound) | Matches any numeric digit |
| @ (At) | Matches any alphabetic character |
| . (Period) | Matches any nonalphanumeric character |
| * (Asterisk) | Matches any string and can be used anywhere in the search string |
| ? (Question mark) | Matches any single character; for example, ?BC matches ABC, 3BC, and so on |
| ~ (Tilde) | Matches anything but the pattern; for example; ~*AB*matches all strings that don't contain AB |
| [ ] | Matches any one of the characters enclosed; for example, [AB]C matches AC and BC |
| [~] | Matches any character not enclosed; for example, [~AB]C matches XC but not AC |
| [-] | Specifies a range for a single character; for example, [A-G]C matches AC, BC, and so on to GC, but not HC |
| ` (Reverse quote) | Reads the next character literally; for example, `~AB matches ~AB |

To search for drawing files that contain a specific word or phrase, use the Search tool in Microsoft® Windows® . You can use the Search tool to find all textual data except text in tables and fields, and Xrefs within drawing files.

**See also:**

"Work with Text Styles" on page 733
"Control the Display of Polylines, Hatches, Gradient Fills, Lineweights, and Text" on page 306

### 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.

Standard toolbar

### To find specified text in a drawing

1. Click Edit menu ➤ Find.
2. In Find Text String, enter the text you want to find.
3. In Search In, select Entire Drawing or click the Select Objects button to select one or more text objects.
4. Click Options to specify the types of text to include in the search, whether to find whole words only, and whether to match the case of the specified text.
5. Click Find.

   The Context area displays the found text with its surrounding context.
6. To zoom to the found text in the drawing, click Zoom To.
7. Click Close.

### To replace text

1. Click Edit menu ➤ Find.
2. In Find Text String, enter the text you want to find.
3. In Search In, select Entire Drawing or click the Select Objects button to select one or more text objects.
4. Click Options to specify the types of text to include in the search, whether to find whole words only, and whether to match the case of the specified text.
5. In Replace With, enter the text with which you want to replace the found text.
6. Click Find.

The Context area displays the found text with its surrounding context.

7 Use *one* of the following methods:

- 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.

The replacement is confirmed in the status area at the bottom of the dialog box.

8 Click Close.

#### 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:

- Click OK on the toolbar.
- Click in the drawing outside the editor.
- Press CTRL + ENTER .

## Change Text with a Leader

You can change both the text component and the leader geometry component of leader objects.

To resize a leader, stretch or scale it. Stretching modifies the offset between the leader endpoint and its annotation. Scaling updates the scale of the selected object only. For example, if you scale the leader, the annotation stays in the same position relative to the leader endpoint but is not scaled itself. Leaders can function as edges for trimming and extending, but they cannot be trimmed or extended.

### To edit leader text

1. Double-click the text you want to edit.

   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.

   You can also select the text and right-click. Click Properties. Edit the text in the Properties palette.

   Any modification to the annotation that changes the annotation position or its attachment point affects the leader's endpoint position. Rotating the annotation causes the leader hook line (if any) to rotate.

# Change Text Scale and Justification

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  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  command converts distances between model space units and paper space units. By using SPACETRANS transparently, you can provide commands with distance entries 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.

For more information about entering commands transparently, see “Enter Commands on the Command Line” on page 52.

### Change the Justification of Text Objects Without Changing Their Location

Use  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.

# Check Spelling

You can check the spelling of all text in your drawing, including

- Single-line text
- Multiline text
- Text within attribute values
- Text within block references and their associated block definitions
- Text within nested blocks

Spelling is checked only in objects in the current selection set. If you enter the All option when selecting objects, spelling is checked in all objects in model space and in all layouts. Spelling is not checked in block definitions of unselected block references or in text within dimensions.

### Switch Dictionaries

The program contains several main dictionaries, which are available in different languages. The main dictionaries use a standard word list that you can customize.

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.

To check spelling in another language, you can change to a different main dictionary. You can also create any number of custom dictionaries and switch to them as needed.

You can change dictionaries in a dialog box or by specifying the dictionary name in the  or  system variable. For a list of the dictionary file names, see DCTMAIN.

### Create and Edit Custom Dictionaries

A custom dictionary is a list of spelling exceptions that you have identified. The files that contain them can be identified by the *.cus* file extension. You can use any ASCII text editor to add and delete words or to combine dictionaries. If a word is preceded by a tilde (~), the word is always flagged as incorrect.

You can also create and edit dictionaries from inside the program.

#### To check spelling

1. Click Tools menu ➤ Spelling.
2. Select the text objects you want to check, or enter **all** to select all text objects.
3. If no mispelled words are found , a message is displayed. If a misspelling is found , the Check Spelling dialog box identifies the misspelled word.
4. 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 custom dictionary, click Add. (This option is not available unless a custom dictionary is specified.)
5. Repeat step 4 for each misspelled word. Click OK or Cancel to exit.

#### To switch dictionaries while checking spelling

1. Click Tools menu ➤ Spelling.
2. Select the text that you want to check and press ENTER .
3. In the Check Spelling dialog box, click Change Dictionaries.
4. Do *one* of the following:
   - To change the main dictionary, select a dictionary from the Main Dictionary list.
   - To change the custom dictionary, select or enter a name under Custom Dictionary.
   - To select from a list of dictionaries, click Browse.
5. Click Apply and Close.

**To create or edit custom dictionaries during a spelling check**

1. Click Tools menu ➤ Spelling.
2. Select the text that you want to check and press ENTER .
3. In the Check Spelling dialog box, click Change Dictionaries.
4. In the Change Dictionaries dialog box, under Custom Dictionary, specify the dictionary you want to edit using one of these methods:
   - To specify a dictionary, enter its name, using the *.cus* file extension.
   - To select from a list of folders, click Browse.
   - To create a new custom dictionary, enter a new name. Use the *.cus* file extension.
5. To add a word, enter it under Custom Dictionary Words. Click Add.
6. To remove a word, select it from the list. Click Delete.
7. Click OK.

# Use an Alternate Text Editor

The default text editor is the In-Place Text Editor, but you can elect to use any alternate editor that saves files in ASCII format.

## Overview of Using an Alternate Text Editor

The default text editor is the In-Place Text Editor, but you can elect to use an alternate editor by specifying the editor with the system variable. You can use any text editor, such as Microsoft Notepad, that saves files in ASCII format.

If you use an alternate text editor for multiline text, you specify the properties of the multiline text object on the command line 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.

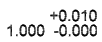
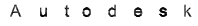
To edit text using an alternate text editor, use the same format codes. To avoid losing format information when you make changes to the text, use the same text editor you used to create the text.

# Format Multiline Text in an Alternate Text Editor

If you use an alternate text editor, you apply formatting by entering format codes. You can underline text, add a line over text, and create stacked text. You can also change color, font, and text height. You can change the spaces between text characters or increase the width of the characters themselves. To apply formatting, use the format codes shown in the following table.

**Format codes for paragraphs**

| Format code | Purpose | Enter this … | To produce this … |
|---|---|---|---|
| \0...\o | Turns overline on and off | Autodesk \OAutoCAD\o | Autodesk AutoCAD |
| \L...\l | Turns underline on and off | Autodesk \LAutoCAD\l | Autodesk AutoCAD |
| \~ | Inserts a nonbreaking space | Autodesk AutoCAD\~LT | Autodesk AutoCAD LT |
| \\ | Inserts a backslash | Autodesk \\AutoCAD | Autodesk \AutoCAD |
| \{...\} | Inserts an opening and closing brace | Autodesk \{AutoCAD\} | |
| \C*value;* | Changes to the specified color | Autodesk \C2;AutoCAD | Autodesk AutoCAD |
| *File name*; | Changes to the specified font file | Autodesk \Ftimes; AutoCAD | Autodesk AutoCAD |
| \H*value*; | Changes to the text height specified in drawing units | Autodesk \H2;AutoCAD | Autodesk AutoCAD |
| \H*valuex*; | Changes the text height to a multiple of the current text height | Autodesk \H3x;AutoCAD | Autodesk AutoCAD |

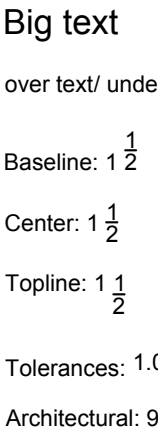
| Format codes for paragraphs | | | |
|---|---|---|---|
| **Format code** | **Purpose** | **Enter this ...** | **To produce this ...** |
| \S...^...; | Stacks the subsequent text at the /, #, or ^ symbol | 1.000\S+0.010^-0.000; |  |
| \T*value*; | Adjusts the space between characters, from .75 to 4 times | \T2;Autodesk |  |
| \Q*angle*; | Changes obliquing angle | \Q20;Autodesk |  |
| \W*value*; | Changes width factor to produce wide text | \W2;Autodesk |  |
| \A | Sets the alignment value; valid values: 0, 1, 2 (bottom, center, top) | \A1;1\S1/2 | |
| \P | Ends paragraph | Autodesk\PAutoCAD | |

Braces can be nested up to eight levels deep.

You can also use control codes to add special characters, such as tolerance or dimensioning symbols. See .

**Example: Formatting Text in an Alternate Text Editor**

This example describes how the text in the following illustration was created.




Each line below was entered in an alternate text editor:

{{\H1.5x; Big text} \A2; over text\A1;/\A0; under text}\P

{\A0;Baseline: 1 \S1/2;}\P

{\A1;Center: 1 \S1/2;}\P

{\A2;Topline: 1 \S1/2;}\P

{Tolerances: \A1;1.000\H.75x;\S+0.010^-0.000;}\P

{Architectural: 9-{\H.666x;\A2;11\A1;/\A0;16}\A2;"}\P

**To specify an alternate text editor**

1. At the Command prompt, enter **mtexted**.
2. At the prompt do *one* of the following:
   - Enter the path and name of the executable file for the ASCII text editor that you want to use to create or edit multiline text.
   - Enter **internal** to restore the text editor.

**To create multiline text in an alternate text editor**

1. To specify a text editor, at the Command prompt, enter **mtexted**. Then enter the path of the editor you want to use.

2. Click Draw menu ➤ Text ➤ 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.

Draw toolbar

# Tables

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A table is a rectangular array of cells that contain annotation, primarily text but also blocks. Tables appear in many different forms on many of the sheets that make up drawing sets. In the AEC industry, tables are often referred to as “schedules” and contain information about the materials needed for the construction of the building being designed. In the manufacturing industry, they are often referred to as “BOM” (bills of materials).

The table object creates a table of any size that can be used for any purpose, including as a list or index to a set of drawing sheets to be published.

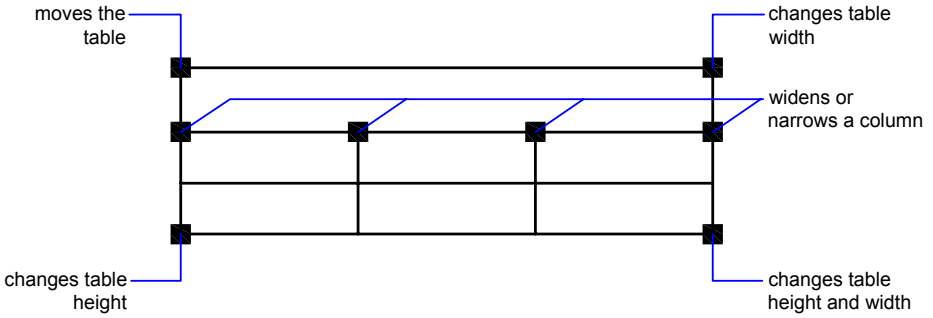
## In this chapter

- Create and Modify Tables
- Work with Table Styles
- Add Text and Blocks to Tables
- Use Formulas in Table Cells

# Create and Modify Tables

A table is an object that contains data in rows and columns. When you create a table object, you first create an empty table, and then you add content to the cells in the table.

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, the rows or columns change proportionally. When you change the width of a column, the table widens or narrows to accommodate the change. To maintain the table width, press CTRL while you use a column grip.

### 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.

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.

With a cell selected, 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, including changes made in the Property palette.

### 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 or block content and character formatting is ignored.

**See also:**

"Add Text and Blocks to Tables" on page 765

### To create a table

1. Click Draw menu ➤ Table.
2. In the Insert Table dialog box, select a table style from the list, or click the [...] button to create a new table style.
3. Select an insertion method:
   - Specify an insertion point for the table.
   - Specify a window for the table to fit.
4. Set the number of columns and the column width.

   If you used the window insertion method, you can select the number of columns or the column width, but not both.
5. Set the number of rows and the row height.

   If you used the window insertion method, the number of rows is determined by the size of the window you specified and the row height.
6. Click OK.

### To use grips to modify a table

1. Click a gridline to select the table.
2. Use one of the following grips:
   - **Upper-left grip**. Moves the table.
   - **Upper-right grip**. Changes the width of the table and all columns proportionally.
   - **Lower-left grip**. Changes the height of the table and all rows proportionally.
   - **Lower-right grip**. Changes both the height and the width of the table and changes the columns and rows proportionally.
   - **Column grip** (at the top of the column head row). Changes the width of the column to the left of the grip and widens or narrows the table to accommodate the change.

- **CTRL + a column grip**. Widens or narrows adjacent columns without changing the width of the table.

The minimum column width is the width of a single character. The minimum row height for an empty table is the height of a line of text plus the cell margins.

3 Press ESC to remove selection.

### To use grips to modify cells in a table

1 Select one or more cells in a table that you want to change using one of the following methods:

- Click inside a cell.
- Hold down SHIFT and click inside another cell to select those two cells and all the cells between them.
- Click inside the selected cell, drag to the cells you want to select, and release.

2 To change the row height of the selected cell, drag the top or bottom grip.

If more than one cell is selected, the row height changes equally for each row.

3 To change the column width of the selected cell, drag the left or right grip.

If more than one cell is selected, the column width changes equally for each column.

4 To merge selected cells, right-click. Click Merge Cells.

If cells in more than one row or column are selected, you can merge by row or by column.

5 Press ESC to remove selection.

### To use the Properties palette to modify a table

1 Click a gridline to select the table.

2 Click Tools menu ➤ Properties.

3 In the Properties palette, click the value you want to change and enter or select a new value.

The property is changed in the selected table.

4 Move the cursor outside the Properties palette, and press ESC to remove selection.

### To change column width or row height in a table

1 Click inside a table cell in the column or row you want to change.

Hold down SHIFT and click inside another cell to select those two cells and all the cells between them.

2 Click Tools menu ➤ Properties.

3 In the Properties palette, under Cell, click the cell width or the cell height value and enter a new value.

4 Press ESC to remove selection.

### To add columns or rows to a table

1 Click inside a table cell where you want to add a column or a row.

You can select more than one cell to add more than one column or row.

2 Right-click and use one of the following options:

- **Insert Column ➤ Right**. Inserts a column to the right of the selected cell.
- **Insert Column ➤ Left**. Inserts a column to the left of the selected cell.
- **Insert Row ➤ Above**. Inserts a row above the selected cell.
- **Insert Row ➤ Below**. Inserts a row below the selected cell.

3 Press ESC to remove selection.

### To merge cells in a table

1 Select the cells in a table that you want to merge using one of the following methods:

- Select a cell, and hold down SHIFT and click inside another cell to select those two cells and all the cells between them.
- Click inside a selected cell, drag to the cells you want to select, and release.

The resulting merged cell must be rectangular.

2. Right-click. Click Merge Cells. If you want to create more than one merged cell, use one of the following options:

   - **By Row:** Merges the cells horizontally by removing the vertical gridlines and leaving the horizontal gridlines intact.
   - **By Column:** Merges the cells vertically by removing the horizontal gridlines and leaving the vertical gridlines intact.

3. Start typing to enter text in the new merged cell, or press ESC to remove selection.

#### To delete columns or rows in a table

1. Click inside a table cell in the column or row that you want to delete.

   Hold down SHIFT and click inside another cell to select those two cells and all the cells between them.

2. Right-click and use one of the following options:

   - **Delete Columns**. Deletes the columns you specified.
   - **Delete Rows**. Deletes the rows you specified.

3. Press ESC to remove selection.

#### To export a table

1. At the Command prompt, enter **tableexport**.
2. Select the table that you want to export.

   A standard file selection dialog box is displayed.

3. Enter a file name and select a location for the file.

   Table data is exported in the comma-separated (CSV) file format. All table and text formatting is lost.

With a table selected, right-click. Click Export.

# 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.

The table style can specify formatting for rows. In the STANDARD table style, for example, the first row is a title row that consists of a row of merged cells

with text that is centered. The second row is for column heads, and all the other rows are data rows.

The table style can specify different justification and appearance for the text and gridlines in each type of row. For example, a table style can specify larger text for the title row or center alignment for the column heads row and left alignment for the data rows.

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 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 preview image in 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 table 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.

**See also:**

"Work with Text Styles" on page 733

### To define or modify a table style

1. Click Format menu ➤ Table Style.
2. In the Table Style dialog box, click New.
3. In the Create New Table Style dialog box, enter a name for the new table style, select a table style in Start With to provide default settings for the new table style, and click Continue.
4. In the New Table Style dialog box, set the following options for the whole table:
   - **Table Direction**. Select Down or Up. Up creates a table that reads from the bottom up; the title row and the column heads are at the bottom of the table.
   - **Cell Margins**. Enter a value for the vertical and horizontal spacing between the border of the cell and the cell content. The default setting is one-third of the text height in the data rows, and the maximum is the text height in the data rows.
5. To set the appearance of the data cells, the column head cells, or the title cell, click the appropriate tab.

6 If you want to include a title row or a header row, on the Title tab or the Column Heads tab (as appropriate), select or clear the following options:

- **Include Title Row**. When this option is selected, the first row in the table is all one cell that has the appearance that you set on the Title tab.
- **Include Header Row**. When this option is selected, the first row in each column is a column heads row that has the appearance that you set on the Column Heads tab.

When both options are selected, the first row of the table is a title row, and the second row is a column heads row.

7 On any of the tabs, use the following cell properties options to control the appearance of the cell contents:

- **Text Style**. Select a text style, or click the [...] button to open the Text Style dialog box and create a new text style.
- **Text Height**. Enter a height for the text. This option is available only when the selected text style has a text height of 0. (The default text style, STANDARD, has a 0 text height.) If the selected text style specifies a fixed text height, this option is unavailable.
- **Text Color**. Select a color, or click Select Color to display the "Select Color Dialog Box"Select Color dialog box.
- **Fill Color**. Select None or a background color, or click Select Color to display the "Select Color Dialog Box"Select Color dialog box.
- **Alignment**. Specify an alignment for cell contents. Center refers to horizontal alignment; Middle refers to vertical alignment.

8 Use the following border properties options to control the appearance of the gridlines:

- **Border display buttons**. Click a button to apply lineweight and color properties to all borders of the cell, the outside border, the inside borders (not available on the Title tab), no borders, or the bottom border. The preview in the dialog box updates to show the effect.
- **Grid Lineweight**. Enter the lineweight to be used for borders that are displayed. If you use a heavy lineweight, you may have to change the cell margins so that the text can be seen.
- **Grid Color**. Select a color to be used for borders that are displayed, or click Select Color to display the "Select Color Dialog Box"Select Color dialog box.

9 Click another tab to continue defining the table style, or click OK to exit the dialog box.

**To apply a new table style to a table**

1 Click a gridline to select the table.

2 Click Tools menu ➤ Properties.

3 In the Properties palette, under Table, click the Table Style value and select a table style.

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.

**To change the table style that is applied to new tables**

1 Click Format menu ➤ Table Style.

2 In the Table Style dialog box, select a table style. Click Set Current.

3 Click Close.

The current table style is applied to new tables that you create.

# Add Text and Blocks to Tables

The data in a table cell can be text or a block.

When the 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.

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.

Inside the cell, the arrow keys move the cursor. Use the toolbar and the shortcut menu to format text, import text, or make other changes to the text in the cell.

**See also:**

"Use Fields in Text" on page 725
"Create Multiline Text" on page 701

### To enter text in a table

1. Click inside a table cell, and start to enter text.

   The Text Formatting toolbar is displayed.
2. Use the arrow keys to move the cursor in text inside a cell.
3. To create a line break within a cell, press ALT + ENTER .
4. To override the text style specified in the table style, click the arrow next to the Text Style control on the toolbar and select a new text style.

   The text style you select is applied to the text in the cell and any new text that you enter in the cell.
5. To override the formatting in the current text style, first select text as follows:

   - To select one or more characters, click and drag the pointing device over the characters.
   - To select a word, double-click the word.
   - To select all the text in the cell, triple-click in the cell. (You can also right-click. Click Select All.)
6. On the toolbar, make format changes as follows:

   - To change the font of the selected text, select a font from the list.
   - To change the height of the selected text, enter a new value in the Height box.
   - To format text in a TrueType font with boldface or italics, or to create underlined text for any font, click the corresponding button on the toolbar. SHX fonts do *not* support boldface or italics.
   - To apply color to selected text, choose a color from the Color list. Click Other to display the "Select Color Dialog Box"Select Color dialog box.
7. Use the keyboard to move from cell to cell:

   - Press TAB to move to the next cell. In the last cell of the table, press TAB to add a new row.
   - Press SHIFT + TAB to move to the previous cell.

- When the cursor is at the beginning or end of text in a cell, use the arrow keys to move the cursor to adjacent cells. You can also use CTRL + an arrow key.
- When text is highlighted in a cell, press an arrow key to remove selection and move the cursor to the beginning or end of the text in the cell.
- Press ENTER to move down one cell.

8 To save your changes and exit, click OK on the toolbar or press CTRL + ENTER .

### To change the properties of cells in a table

1 Click inside the table cell you want to change.

Hold down SHIFT and click inside another cell to select those two cells and all the cells between them.

2 Use one of the following methods:

- To change one or more properties, in the Properties palette, click the value you want to change and enter or select a new value.
- To restore the default properties, on the shortcut menu, 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 Right-click. 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 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. Right-click. Click Cell Borders.
3. In the Cell Border Properties dialog box, select a lineweight and a color.

   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 edit text in a table cell

1. Double-click inside the cell whose text you want to edit, or select the cell and click Edit Cell on the shortcut menu.
2. Use the Text Formatting toolbar or the shortcut menu to make changes.
3. To save the changes and exit, click OK on the toolbar, press CTRL + ENTER , or click outside the cell.
4. To remove selection from the table, press ESC .

#### To insert a block in a table cell

1. Click inside the table cell to select it and right-click. Click Insert Block.
2. In the Insert dialog box, select a block from the list of blocks in the drawing, or click Browse to find a block in another drawing.
3. Specify the following properties for the block:

   - **Cell Alignment**. Specifies alignment for the block in the table cell. The block is middle-, top-, or bottom-aligned with respect to the top and bottom borders of the cell. The block is center-, left-, or right-aligned with respect to the left and right borders of the cell.
   - **Scale**. Specifies the scale for the block reference. Enter a value or select AutoFit to scale the block to fit in the selected cell.
   - **Rotation Angle**. Specifies a rotation angle for the block.
4. Click OK.

   If the block has attributes attached, the Edit Attributes dialog box is displayed.

### To insert a field in a table cell

1. Double-click inside the table cell.
2. Right-click. Click Insert Field, or press CTRL + F .
3. In the Field dialog box, select a category in the Field Category list to display the field names in that category.
4. Select a field.
5. Select the format or other options available for that field.
6. Click OK.

# 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 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.

### 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.

### 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  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:**

"Use Fields in Text" on page 725

### To sum the values in a range of table cells

1. Select the table cell where you want to place the formula by clicking inside it.
2. Right-click. Click Insert Formula ➤ Sum.

   The following prompt is displayed:

Select first corner of table range:

3. Click inside the first cell in the range.

   The following prompt is displayed:

Select second corner of table range:

4. Click inside the last cell in the range.

   The In-Place Text Editor opens and displays the formula in the cell.
5. Edit the formula, if necessary.
6. To save your changes and exit the editor, use one of the following methods:

   - Click OK on the toolbar.
   - Click in the drawing outside the editor.
   - Press CTRL + ENTER .

   The cell displays the sum of the values in the range of cells. Empty cells and cells that do not resolve to a numeric value are ignored.

**To average the values in a range of table cells**

1. Select the table cell where you want to place the formula by clicking inside it.
2. Right-click. Click Insert Formula ➤ Average.

   The following prompt is displayed:

Select first corner of table range:

3. Click inside the first cell in the range.

   The following prompt is displayed:

Select second corner of table range:

4. Click inside the last cell in the range.

   The In-Place Text Editor opens and displays the formula in the cell.
5. Edit the formula, if necessary.
6. To save your changes and exit the editor, use one of the following methods:

   - Click OK on the toolbar.
   - Click in the drawing outside the editor.
   - Press CTRL + ENTER .

   The cell displays the average of the values in the range of cells. Empty cells and cells that do not resolve to a numeric value are ignored.

**To count the cells in a table column or a table row**

1. Select the table cell where you want to place the formula by clicking inside it.
2. Right-click. Click Insert Formula ➤ Count.

   The following prompt is displayed:

Select first corner of table range:

3. Click inside the first cell in the range.

   The following prompt is displayed:

Select second corner of table range:

4. Click inside the last cell in the range.

   The In-Place Text Editor opens and displays the formula in the cell.
5. Edit the formula, if necessary.

6 To save your changes and exit the editor, use one of the following methods:

- Click OK on the toolbar.
- Click in the drawing outside the editor.
- Press CTRL + ENTER .

The cell displays the total number of cells in the range.

### To use a cell from a different table in a formula

1 Select the table cell where you want to place the formula by clicking inside it.

2 Right-click. Click Insert Formula ➤ Cell.

The following prompt is displayed:

Select table cell:

3 Click inside the cell in the other table.

The In-Place Text Editor opens and displays the cell address.

4 Enter the rest of the formula.

5 To save your changes and exit the editor, use one of the following methods:

- Click OK on the toolbar.
- Click in the drawing outside the editor.
- Press CTRL + ENTER .

The cell displays the result of the calculation.

### To manually enter a formula in a table cell

1 Double-click inside a table cell.

The In-place Text Editor opens.

2 Enter a formula (a function or an arithmetic expression), as in the following examples:

- **=sum(a1:a25,b1)**. Sums the values in the first 25 rows of column A and the first row in column B.
- **=average(a100:d100)**. Calculates the average of the values in the first 4 columns in row 100.

- **=count(a1:m500)**. Displays the total number of cells in column A through column M in rows 1 through 100.
- **=(a6+d6)/e1**. Adds the values in A6 and D6 and divides the total by the value in E1.

  Use a colon to define a range of cells and a comma for individual cells. A formula must start with an equal sign (=) and can contain any of the following signs: plus (+), minus (-), times (*), divided by (/), exponent (^), and parentheses ().

3. To save your changes and exit the editor, use one of the following methods:
   - Click OK on the toolbar.
   - Click in the drawing outside the editor.
   - Press CTRL + ENTER .

   The cell displays the result of the calculation.

#### To turn off the display of column letters and row numbers for tables

1. On the command line, enter **tableindicator**.
2. At the Enter New Value prompt, enter **0**.

When is set to 1, the In-Place Text Editor displays column letters and row numbers when a table cell is selected.

#### 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 Color Indicator.
3. In the Select Color dialog box, select a color.
4. Click OK.

   The text color, size, and style and the line color are controlled by the settings for column heads in the current table style.

#### To insert a Formula field in a table cell

1. Double-click inside a table cell.
2. In the In-Place Text Editor, right-click. Click Insert Field.
3. In the Field dialog box, Field Category list, select Objects.

4 In Field Names, select Formula.

5 To enter a formula, use any of the following methods one or more times:

- Click Average, Sum, or Count. The Field dialog box closes temporarily. To specify a range, click inside the first and the last cell. The result is appended to the formula.
- Click Cell. The Field dialog box closes temporarily. Select a cell in a table in the drawing. The cell address is appended to the formula.

6 (Optional) Select a format and a decimal separator.

7 Click OK.

8 To save your changes and exit the editor, use one of the following methods:

- Click OK on the toolbar.
- Click in the drawing outside the editor.
- Press CTRL + ENTER .

The cell displays the result of the calculation.

# Dimensions and Tolerances

# 25

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.

## In this chapter


- Understand Basic Concepts of Dimensioning
- Use Dimension Styles
- Set the Scale for Dimensions
- Create Dimensions
- Modify Existing Dimensions
- Add Geometric Tolerances

# 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.

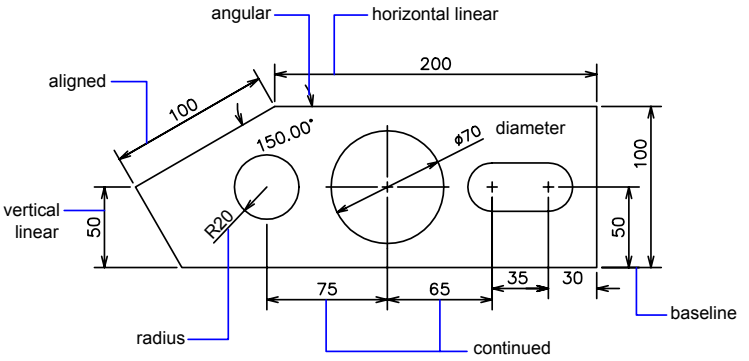
## Overview of Dimensioning

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 and diameter)
- 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.

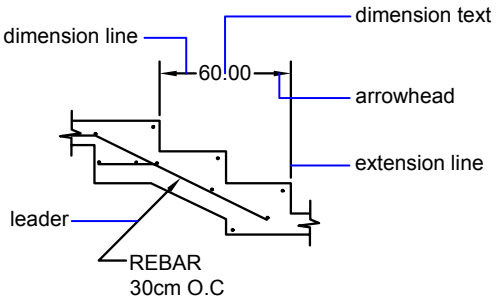
**To create a dimension**

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 the Dimension menu and select a dimension command.
4. Follow the command prompts.

# 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.

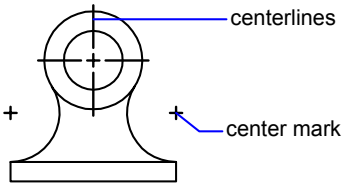
A *dimension line* indicates the direction and extent of a dimension. For angular dimensions, the dimension line is an arc.

*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.




# 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 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 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 command displays the associative and nonassociative elements of a dimension.

### Special Situations and Limitations

You may need to use  to update associative dimensions after panning or zooming with a wheel mouse, after opening a drawing that was modified with an earlier release, or after opening a drawing with external references that have been modified.

Although associative dimensions support most object types that you would expect to dimension, they do not support the following:

- Hatches
- Multiline objects
- 2D solids
- Objects with nonzero thickness

When selecting objects to dimension, make sure that the objects that you select don't include a directly overlapping object that does not support associative dimensioning such as a 2D solid.

Associativity is *not* maintained between a dimension and a block reference if the block is redefined.

Associativity is not maintained between a dimension and a 3D solid if the shape of the 3D solid is modified.

Dimensions created with  are not associative but may be associated individually with DIMREASSOCIATE.

---

**NOTE** In releases prior to AutoCAD 2002, the definitions of associative and nonassociative dimensions were different and were controlled by the DIMASO system variable. The behavior of dimensions is now controlled by the DIMASSOC system variable.

---

For information about working with associative dimensions in combination with previous releases, see "Save Drawings to Previous Drawing File Formats".

**See also:**

"Change Dimension Associativity" on page 833

### 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.

# 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.

# 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 Dimension menu ➤ 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.

Dimension or Styles toolbar

**To list settings for an existing dimension style**

1. Click Dimension menu ➤ 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.

Dimension or Styles toolbar

**To list dimension styles in the current drawing**

1. Click Dimension menu ➤ Style.
2. In the Dimension Style Manager, under List, select All Styles or Style in Use.

Dimension or Styles toolbar

**To compare dimension styles**

1. Click Dimension menu ➤ 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.

Dimension or Styles toolbar

# Control Dimension Geometry

You can control the appearance of dimension lines, extension lines, arrowheads, and center marks.

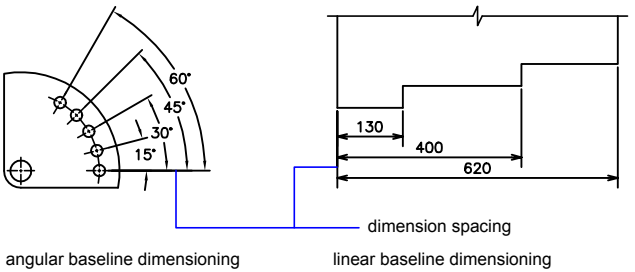
## Control Dimension Lines

You can control dimension line properties including color, lineweight, and spacing.

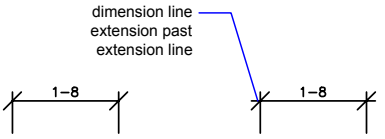
You can control several aspects of a dimension line. You can

- 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 Dimension menu ➤ 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.

Dimension or Styles toolbar

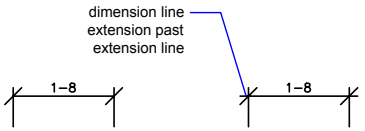
## Control Extension Lines

You can control extension line properties including color, lineweight, overshoot, and offset length.

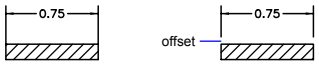
You can control several aspects of the extension lines. You can

- Specify color and lineweight for visual effect and plotting
- Suppress one or both extension lines if they are unnecessary, or if there is not enough space

- Specify how far beyond the dimension line that the extension line extends (overshoot)



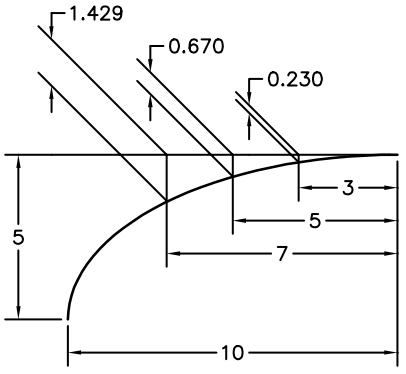

- Control the extension origin offset, the distance between the extension line origin, and the start of the extension line




- Specify a fixed length for extension lines, as measured from the dimension line toward the extension line origin

- Specify a noncontinuous linetype, typically used for centerlines

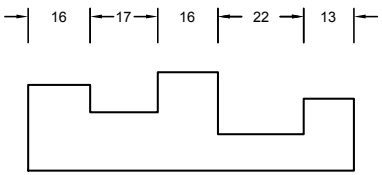
- Modify the angle of the extension lines of a selected dimension to make them oblique



dimensions set to oblique

### Fixed-Length Extension Lines

With the Dimension Style Manager, on the Lines tab, you can specify a dimension style that sets the total length of extension lines starting from the dimension line toward the dimension origin point.



fixed-length extension lines

The extension-line offset distance from the origin will never be less than the value specified by the  system variable.

**See also:**

"Create Dimensions with Oblique Extension Lines" on page 814

### To modify the display of extension lines

1. Click Dimension menu ➤ 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.

Dimension or Styles toolbar

## Control Dimension Arrowheads

You can control the arrowhead symbols in dimensions and leaders including their type, size, and visibility.

You can choose from many standard types of arrowheads, or you can create your own arrowheads. Additionally, you can

- Suppress the display of arrowheads, or use one arrowhead only
- Apply a different type of arrowhead to each end of a dimension line
- Control the size of arrowheads
- Flip the direction of an arrowhead using the dimension shortcut menu

---

**NOTE** Flipped arrowheads maintain their appearance in versions later than AutoCAD 2002. However, if you edit a drawing with flipped arrowheads in a release earlier than AutoCAD 2006, the arrowhead directions will revert to their original orientations.

---

**See also:**

"Customize Arrowheads" on page 788

**To choose an arrowhead**

1 Click Dimension menu ➤ 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.

Dimension or Styles toolbar

#### 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.

## 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 453.

Arrowhead sizing relies on the overall dimension scale factor. When you create a dimension, the block is inserted where the arrowheads would normally go. The object's *X* and *Y* scale factors are set to *arrowhead size x overall scale*. The dimension line is trimmed by *text gap x overall scale* units at each end. To trim the dimension line, the rightmost block is inserted with a zero rotation angle for horizontal dimensioning. The leftmost block is rotated 180 degrees about its insertion point.

If you use paper-space scaling, the scale factor is computed before applying it to the arrowhead size value.

#### To use your own arrowhead symbol

1. Click Dimension menu ➤ 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 Dinemsion Style Manager.

Dimension or Styles toolbar

# Control Dimension Text

You can control the placement of dimension text, arrowheads, and leader lines relative to the dimension and extension lines.

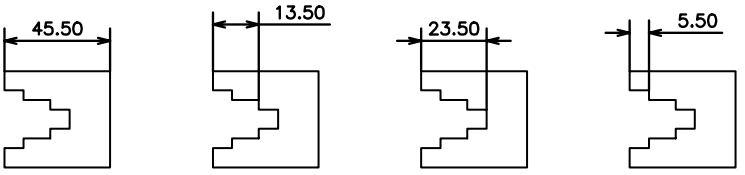
## Fit Dimension Text Within Extension Lines

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.

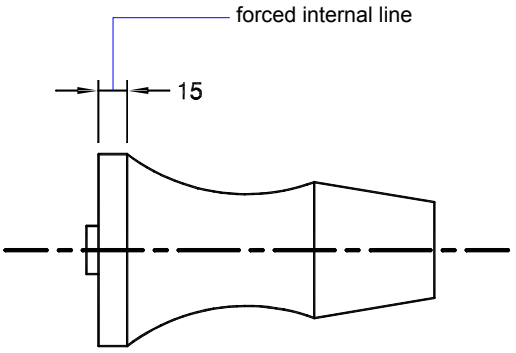
When creating new dimensions, you can choose to place text by entering a coordinate or using the pointing device; this is known as user-defined text placement. Alternatively, the program can compute the text position for you. The options for automatic fitting of text and arrowheads are listed in the 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.



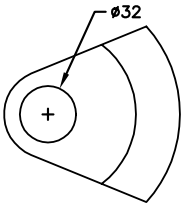

If there is no room for text between the extension lines, you can have a leader line created automatically. This is useful in cases where text outside the extension lines would interfere with other geometry, for example, in continued dimensions. Whether text is drawn to the right or the left of the leader 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.

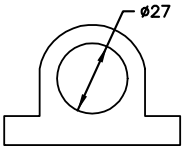



### Fit Diameter Dimension Text

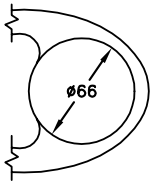
You can draw several different diameter dimensions depending on text placement, horizontal settings on the Text tab, and whether you select the Always Draw Dim Line Between Ext Lines options on the Fit tab.




default horizontal placement-text outside circle, center mark, no forced interior line




user-defined horizontal placement-dimension line and arrows, no center mark, forced interior line




Text and Arrows inside with Inside Horizontal option selected

**To place text within extension lines**

1. Click Dimension menu ➤ 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.

Dimension or Styles toolbar

**To force an internal line and choose a fit option**

1. Click Dimension menu ➤ 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.

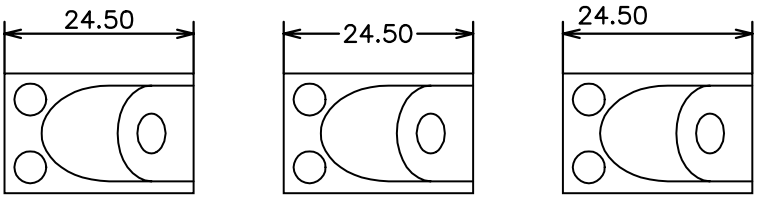
6 Click Close to exit the Dimension Style Manager.

Dimension or Styles toolbar

## Control the Location of Dimension Text

You can locate dimension text manually and specify its alignment and orientation.

The program comes with several justification settings that facilitate compliance with international standards, or you can choose your own location for the text.




text centered above the dimension line

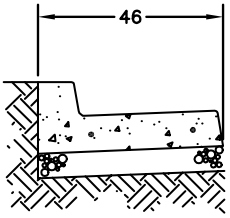
text centered horizontally and vertically

text left-justified above the dimension line

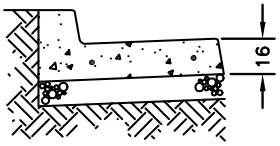
Many of the settings are interdependent. Example images in the Dimension Style Manager are updated dynamically to illustrate how text appears as you change the settings.

### Align Dimension Text

Whether text is inside or outside the extension lines, you can choose whether it is aligned with the dimension line or remains horizontal. The following examples show two combinations of these options.




text inside extension lines
oriented horizontally

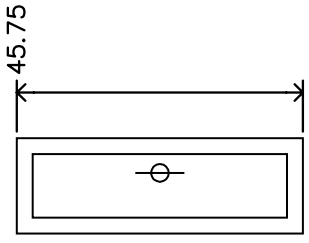



text outside extension lines
aligned with dimension line

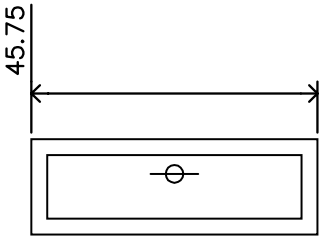
The default alignment is horizontal dimension text, even for vertical dimensions.

### Position Dimension Text Horizontally

The position of the text along the dimension line in relation to the extension lines is referred to as text placement. To place text yourself when you create a dimension, use the Place Text Manually When Dimensioning option on the Fit tab of the Modify/New Dimension Style dialog box. Use the text placement options to automatically place text at the center of the dimension line, at either extension line, or over either extension line.

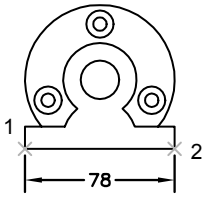



horizontal justification over extension line one, vertical justification of text, Centered over extension line

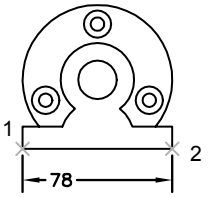



horizontal justification over extension line one, vertical justification of text, Above extension line

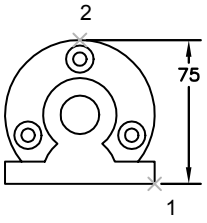
*First* and *second* extension lines are defined by the order in which you specified the extension line origins when you created the dimension. For angular dimensions, the second extension line is counterclockwise from the first. In the following illustrations, 1 is the first extension line origin and 2 the second.




text horizontal and centered along dimension line




text horizontal at first extension line

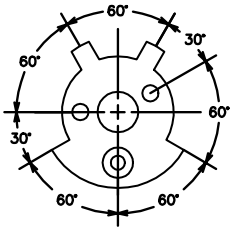



text horizontal at second extension line

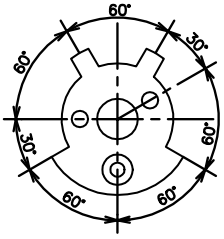
If you place text manually, you can place the dimension text anywhere along the dimension line, inside or outside the extension lines, as you create the dimension. This option provides flexibility and is especially useful when space is limited. However, the Horizontal alignment options provide better accuracy and consistency between dimensions.

### Position Dimension Text Vertically

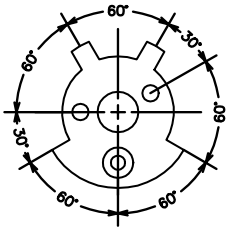
The position of the text relative to the dimension line is referred to as vertical text placement. Text can be placed above or below or centered within the dimension line. In the ANSI standards, centered text usually splits the dimension line. In the ISO standards, it is usually above or outside the dimension line. For example, ISO standards permit angular dimension text to appear in any of the ways shown.




ANSI standard text centered within the dimension line




ISO standard text above the dimension line




text aligned with and centered within the dimension line

Other settings, such as Text Alignment, affect the vertical alignment of text. For example, if Horizontal Alignment is selected, text inside the extension lines and centered within the dimension line is horizontal, as shown in the leftmost illustration above. The text is horizontal even if the dimension line is not itself horizontal.

**To align text with the dimension line**

1. Click Dimension menu ➤ 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.

Dimension or Styles toolbar

**To place text at the second extension line**

1. Click Dimension menu ➤ 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.

Dimension or Styles toolbar

**To place dimension text manually**

1. Click Dimension menu ➤ Style.

2 In the Dimension Style Manager, select the style you want to change. Click Modify.

3 In the Modify Dimension Style dialog box, Fit tab, under Fine Tuning, select Place Text Manually When Dimensioning.

4 Click OK.

5 Click Close to exit the Dimension Style Manager.

As you create dimensions, you can move the text along the dimension line. Use the pointing device or enter coordinates to specify the dimension line and text locations.

Dimension or Styles toolbar

**To place text above the dimension line**

1 Click Dimension menu ➤ 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.

Dimension or Styles toolbar

## 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 733.

### 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

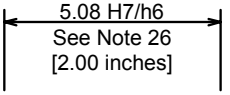
In this example, the primary dimension measurement is 5.08, and the alternate dimension measurement is 2.00. The primary units have the suffix *H7/h6*, and the alternate units have the suffix *inches*.

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 Dimension menu ➤ 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.

Dimension or Styles toolbar

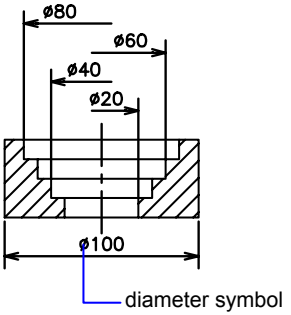
# Control Dimension Values

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).



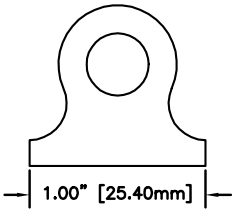

These settings are available on the Primary Units tab of the Dimension Style Manager.

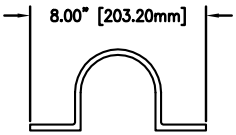
### Control the Display of Alternate Units

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 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 Dimension menu ➤ 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.

Dimension or Styles toolbar

**To add and format alternate units**

1. Click Dimension menu ➤ 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.

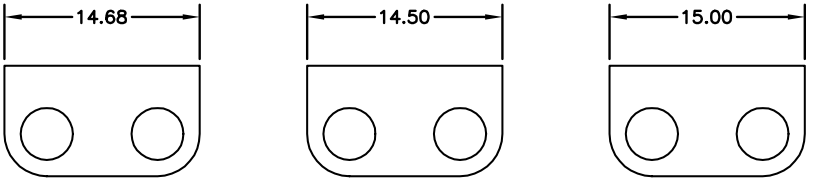
6 Click Close to exit the Dimension Style Manager.

Dimension or Styles toolbar

## Round Off Dimension Values

You can round off the numeric values in dimensions and lateral tolerances.

You can round off all dimension values except those for angular dimensions. For example, if you specify a round-off value of 0.25, all distances are rounded to the nearest 0.25 unit. The number of digits displayed after the decimal point depends on the precision set for primary and alternate units and lateral tolerance values.




round-off value set to default (0)

round-off value set to default .25

round-off value set to default 1

**To round off dimension values**

1 Click Dimension menu ➤ 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.

Dimension or Styles toolbar

## Suppress Zeros in Dimensions

You can suppress leading and trailing zeros in the numeric portion of dimension text.

If you suppress leading zeros in decimal dimensions, 0.500 becomes .500. If you suppress trailing zeros, 0.500 becomes 0.5. You can suppress *both* leading and trailing zeros so that 0.5000 becomes .5 and 0.0000 becomes 0.

The table shows the effect of selecting each option and provides examples of the architectural units style. If feet are included with a fractional inch, the number of inches is indicated as zero, no matter which option you select. Thus, the dimension 4'-3/4" becomes 4'-0 3/4".

| Zero suppression for feet and inches | | | | | |
|---|---|---|---|---|---|
| **Option** | **Effect** | **Examples** | | | |
| No options selected | Includes zero feet and zero inches | 0'-0 1/2" | 0'-6" | 1'-0" | 1'-0 3/4" |
| 0 Inches selected | Suppresses zero inches (includes zero feet) | 0'-0 1/2" | 0'-6" | 1' | 1'-0 3/4" |
| 0 Feet selected | Suppresses zero feet (includes zero inches) | 1/2" | 6" | 1'-0" | 1'-0 3/4" |
| 0 Feet and 0 Inches selected | Suppresses zero feet and zero inches | 1/2" | 6" | 1' | 1'-0 3/4" |

**To suppress zeros in dimension values**

1 Click Dimension menu ➤ 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:

- **Leading.** 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.

Dimension or Styles toolbar

## Display Lateral Tolerances

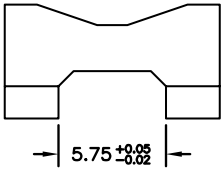
Lateral tolerances are values indicating the amount a measured distance can vary. You can control whether lateral tolerances are displayed and you can choose from several styles of lateral tolerances.

A lateral tolerance specifies the amount by which a dimension can vary. By specifying tolerances in manufacturing, you can control the degree of accuracy needed for a feature. A feature is some aspect of a part, such as a point, line, axis, or surface.

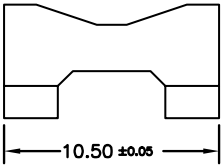
You can apply tolerances directly to a dimension by appending the tolerances to the dimension text. These dimension tolerances indicate the largest and smallest permissible size of the dimension. You can also apply geometric tolerances, which indicate deviations of form, profile, orientation, location, and runout.

Lateral tolerances can be specified from theoretically exact measurements. These are called basic dimensions and have a box drawn around them.

If the dimension value can vary in both directions, the plus and minus values you supply are appended to the dimension value as deviation tolerances. If the deviation tolerance values are equal, they are displayed with a ± sign and they are known as symmetrical. Otherwise, the plus value goes above the minus value.

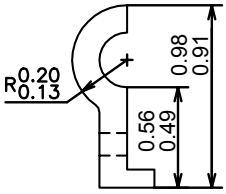



deviation tolerance



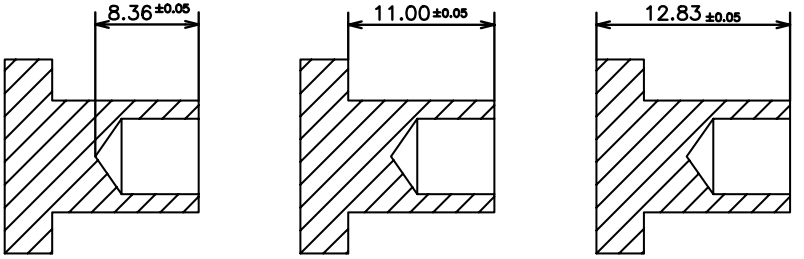

symmetrical deviation tolerance

If the tolerances are applied as limits, the program uses the plus and minus values you supply to calculate a maximum and minimum value. These values replace the dimension value. If you specify limits, the upper limit goes above the lower.




#### Format Lateral Tolerances

You can control the vertical placement of tolerance values relative to the main dimension text. Tolerances can align with the top, middle, or bottom of the dimension text.




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 835

**To specify methods for lateral tolerances**

1. Click Format menu ➤ 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.

Dimension or Styles toolbar

**To align and suppress zeros in tolerance values**

1. Click Dimension menu ➤ 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.

Dimension or Styles toolbar

# 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.

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  system variable to the inverse of the intended plot scale. For example, if the plot scale is 1/4, set  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:  and DIMSCALE do not need to be changed from their default value of 1.0000.

---

**NOTE** When you dimension model space objects in paper space using associative dimensions, dimension values for the display scale of each viewport are automatically adjusted. This adjustment is combined with the current setting for DIMLFAC and is reported by the LIST command as a dimension style override. For nonassociative dimensions, you must set DIMLFAC  manually.

---

**See also:**

"Draw, Scale, and Annotate in Model Space" on page 198
"Scale Views in Layout Viewports" on page 216

**To set the overall dimension scale**

1. Click Dimension menu ➤ 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.

Dimension or Styles toolbar

**To set the dimension scale for model space dimensions in layouts**

1. Click Dimension menu ➤ 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 (Paperspace).
4. Click OK.
5. Click Close to exit the Dimension Style Manager.

Dimension or Styles toolbar

**To set dimension scale for creating dimensions in a layout**

1. Click a layout tab to switch to paper space.
2. To create dimensions in paper space with the correct model space dimension values, use object snap modes to snap to points in model space from paper space or select the objects directly.

   The DIMLFAC system variable can be changed if you need to convert the linear dimension values between the imperial and metric measurement systems.

# Create Dimensions

You can create all of the standard types of dimensions.

# Create Linear Dimensions

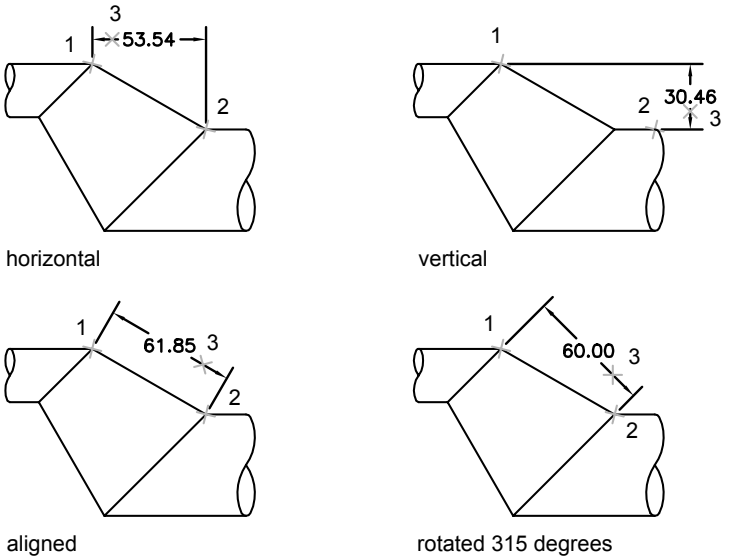
You can create linear dimensions with horizontal, vertical, and aligned dimension lines. These linear dimensions can also be stacked, or they can be created end to end.

## 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.

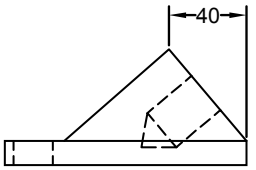



As you create linear dimensions, you can modify the content of the text, the angle of the text, or the angle of the dimension line.

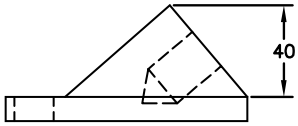
## Create Horizontal and Vertical Dimensions

You can create dimensions using only the horizontal or vertical components of the locations or objects that you specify.

The program automatically applies a horizontal or vertical dimension according to the extension line origins that you specify or the location where you select an object; however, you can override this as you create the dimension by specifying that a dimension be horizontal or vertical. For example, in the following illustration, a horizontal dimension is drawn by default unless you specify a vertical one.




horizontal dimension created by default




vertical dimension specified

**To create a horizontal or vertical dimension**

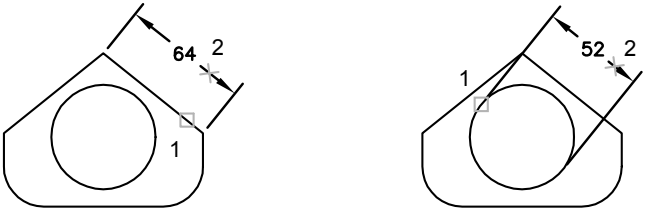
1. Click Dimension menu ➤ Linear.
2. Press ENTER to select the object to dimension, or specify the first and second extension line origins.
3. Before specifying the dimension line location, you can override the dimension direction and edit the text, the text angle, or the dimension line angle:
   - To 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.

Dimension toolbar

## 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.



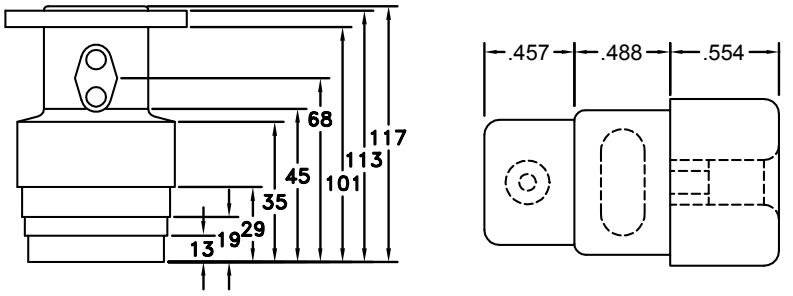

**To create an aligned dimension**

1. Click Dimension menu ➤ Aligned.
2. Press ENTER to select the object to dimension, or specify the first and second extension line origins.
3. Before specifying the dimension line location, you can edit the text or change the text angle. Editing within or overwriting the brackets (<>) changes or removes the dimension value calculated by the program. Adding text before or after the brackets appends text before or after the dimension value.
   - To edit the text using multiline text, enter **m** (multiline text). In the In-place Text Editor, revise the text. Click OK.
   - To edit the text using single-line text, enter **t** (Text). Revise the text on the command line and press ENTER .
   - To rotate the text, enter **a** (Angle). Then enter the text angle.
4. Specify the dimension line location.

Dimension toolbar

# 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 Dimension menu ➤ 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.

Dimension toolbar

**To create a continued linear dimension**

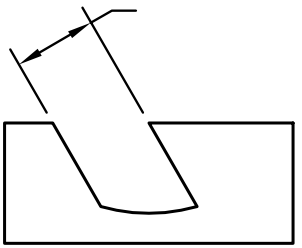
1. Click Dimension menu ➤ Continue.

   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.

Dimension toolbar

## 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 Dimension menu ➤ 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.

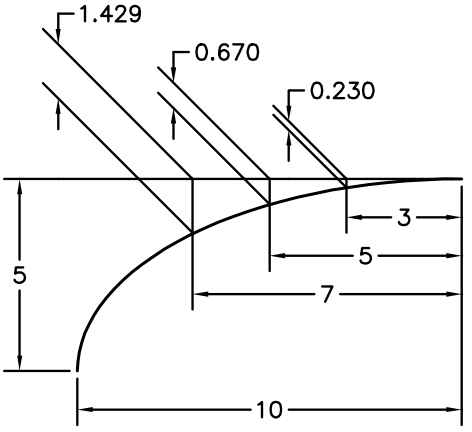
Dimension toolbar

## 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.




dimensions set to oblique

**To make extension lines oblique**

1 Click Dimension menu ➤ Oblique.

2 Select the dimension.

3 Enter a value for the angle of obliqueness, or specify two points.

Dimension toolbar

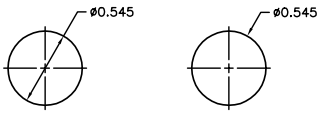
# Create Radial Dimensions

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:

- measures the radius of an arc or circle, and displays the dimension text with the letter *R* in front of it.

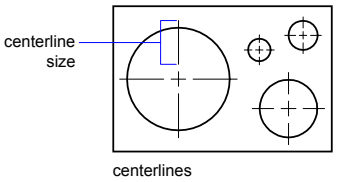
- measures the diameter of an arc or circle, and displays the dimension text with the diameter symbol in front of it.




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 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 command.



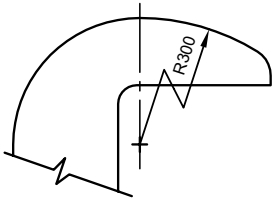

You can control the size and visibility of centerlines and center marks on the Modify Dimension Style dialog box, Symbols and Arrows tab, under Center Marks. You can also access this setting with the  system variable.

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.

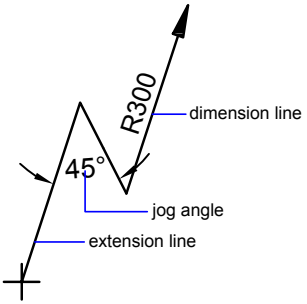
The size of the center mark is the distance from the center of the circle or arc to the end of the center mark.

### Create Jogged Radius Dimensions

With , 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.

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**See also:**

"Fit Dimension Text Within Extension Lines" on page 789

### To create a diameter dimension

1. Click Dimension menu ➤ 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.

Dimension toolbar

**To create a radius dimension**

1. Click Dimension menu ➤ 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.

Dimension toolbar

**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.

Dimension toolbar

**To create centerlines automatically with radial dimensions**

1. Click Dimension menu ➤ 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 Center Marks, select Line in the Type box. The example area reflects your selection.

4 In the Size box, enter the centerline size.

Dimension or Styles toolbar

**To create centerlines or center marks on an arc or circle**

1 Click Dimension menu ➤ 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 Center Marks, select Line in the Type box. The example area reflects your selection.

4 In the Size box, enter the centerline size.

5 Click Dimension menu ➤ Center Mark.

6 Select an arc or a circle.

Dimension toolbar

# 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

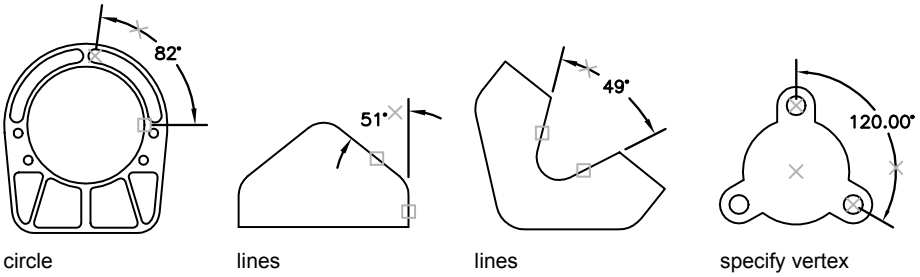
than 180 degrees, use grip editing to stretch the location of the extension line of an existing baseline or continued dimension.

### 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.

### To create an angular dimension

1. Click Dimension menu ➤ 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:

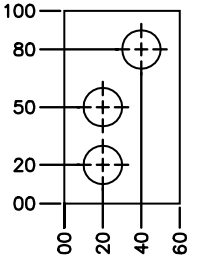
- To edit the dimension text content, enter **t** (Text) or **m** (multiline text). Editing within or overwriting the brackets (<>) changes or removes the calculated dimension value. Adding text before or after the brackets appends text before or after the dimension value.
- To edit the dimension text angle, enter **a** (Angle).

4 Specify the dimension line arc location.

Dimension toolbar

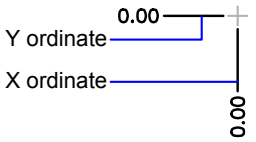
# Create Ordinate Dimensions

Ordinate dimensions measure the perpendicular distance from an origin point called the *datum* to a dimensioned feature, such as a hole in a part. These dimensions prevent escalating errors by maintaining accurate offsets of the features from the datum.

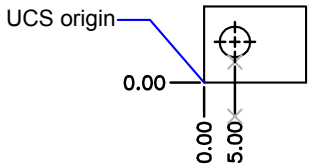



ordinate dimensions

Ordinate dimensions consist of an *X* or *Y* value with a leader line. *X*-datum ordinate dimensions measure the distance of a feature from the datum along the *X* axis. *Y*-datum ordinate dimensions measure the distance along the *Y* axis. If you specify a point, the program automatically determines whether it is an *X*- or *Y*-datum ordinate dimension. This is called an automatic ordinate dimension. If the distance is greater for the *Y* value, the dimension measures the *X* value. Otherwise, it measures the *Y* value.




The program uses the absolute coordinate value of the current UCS to determine the ordinate values. Before creating ordinate dimensions, you typically reset the UCS origin to coincide with the datum.




The dimension text is aligned with the ordinate leader line, regardless of the text orientation defined by the current dimension style. You can accept the default text or supply your own.

**To create ordinate dimensions**

1. Click Dimension menu ➤ Ordinate.
2. If straight ordinate leaders are required, turn Ortho mode on.
3. At the Select Feature Location prompt, specify a point location.
4. Enter **x** (X Datum) or **y** (Y Datum).

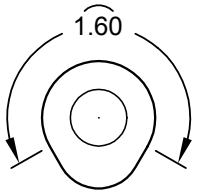
   You can skip this step by making sure that the ordinate leader endpoint is close to vertical for an *X* datum or close to horizontal for a *Y* datum.
5. Specify the ordinate leader endpoint.

Dimension toolbar

# Create Arc Length Dimensions

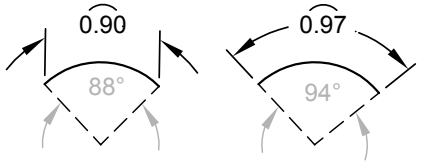
Arc length dimensions measure the distance along an arc or polyline arc segment.

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 Dimension menu ➤ Arc Length.
2. Select an arc or polyline arc segment.
3. Specify the dimension line location.

Dimension toolbar

# Modify Existing Dimensions

You can modify all components of the existing dimension objects in a drawing either individually or by using dimension styles.

## Apply a New Dimension Style to Existing Dimensions

You can modify existing dimensions by applying a different dimension style. If you make changes to a dimension style, you can choose whether to update the dimensions associated with that dimension style.

When you create a dimension, the current dimension style is associated with that dimension. The dimension retains this dimension style unless you apply a new dimension style to it or set up dimension style overrides.

You can modify existing dimensions by applying a different dimension style. If you make changes to a dimension style, you can choose whether to update the dimensions associated with that dimension style.

You can restore an existing dimension style or apply the current dimension style, including any dimension style overrides, to selected dimensions.

**To apply the current dimension style to existing dimensions**

1. Click Dimension menu ➤ Update.
2. Select the dimensions to update to the current dimension style.
3. Press ENTER.

Dimension toolbar

**To restore a dimension style**

1. Click Dimension menu ➤ Style.

2 In the Dimension Style Manager, select the dimension style to restore. Click Set Current.

3 Click Close.

Dimension toolbar

**Alternate**

- On the Styles toolbar, click the arrow in the Dimension Styles control and select a dimension style.

# 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 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 on the command line. 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 on the Command Line

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: **dimlinear**
Specify first extension line origin or <select object>: **dimclrd**
Enter new value for dimension variable <byblock>: **5**
Specify first extension line origin <select object>: *Specify the first extension line origin, or select an object to dimension*

#### To set up dimension style overrides

1. Click Dimension menu ➤ 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.

Dimension or Styles toolbar

**To apply dimension style overrides**

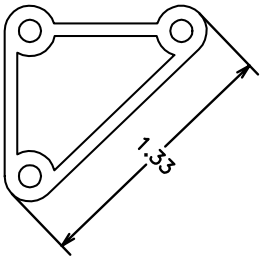
1. Click Dimension menu ➤ Style.
2. In the Dimension Style Manager, click Override.
3. In the Override Current Style dialog box, enter the style overrides. Click OK.

   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.

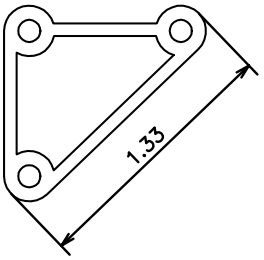
Dimension or Styles toolbar

# Modify Dimension Text

Once you've created a dimension, you can rotate the existing text or replace it with new text. You can move the text to a new location or back to its home position, which is the position defined by the current dimension style. In the following illustration, the home position is above and centered on the dimension line.



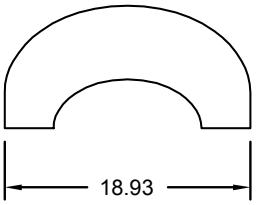

dimension text rotated



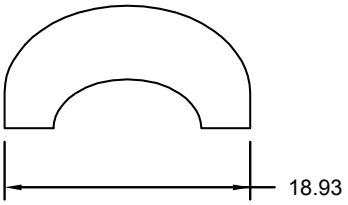

dimension text moved back to home position

When you rotate or replace dimension text, you specify the change first, for example, rotating the text to be at an angle. When you move dimension text, you select a single dimension to move.

You can move dimension text to the left, right, or center along the dimension line or to any position inside or outside the extension lines. A quick and simple way to do this is by using grips. If you move text up or down, the current vertical alignment of the text relative to the dimension line is not changed, so the dimension and extension lines are modified accordingly. The following illustration shows the result of moving text down and to the right. The text remains centered vertically in relation to the dimension line.




text centered vertically on the dimension line




result of moving text to the right and outside the extension lines

**See also:**

"Control Dimension Text" on page 789

**To rotate dimension text**

1. Click Dimension menu ➤ Align Text ➤ Angle.
2. Select the dimension to edit.
3. Enter the new angle for the text.

Dimension toolbar

**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.

Dimension toolbar

**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.

Text toolbar

**To move text to the left side of the dimension line**

1. Click Dimension menu ➤ Align Text ➤ Left.
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.

Dimension toolbar

**To set dimension line spacing for baseline and continued dimensions**

1. Click Dimension menu ➤ 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.

Dimension or Styles toolbar

**To change the extension origin offset**

1. Click Dimension menu ➤ 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.

Dimension or Styles toolbar

# 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.

---

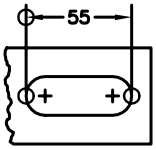
**NOTE** The command line displays a warning message if a dimension is disassociated.

---

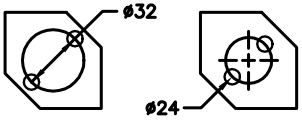
#### Modify Nonassociative Dimensions

For nonassociative dimensions, when you edit dimensioned objects, you must include the relevant dimension definition points in the selection set, or the dimension is not updated. Definition points determine the dimension location. For example, to stretch a dimension, you must include the appropriate definition points in the selection set. You can easily include them by turning on grips and selecting the object so that the grips are highlighted.

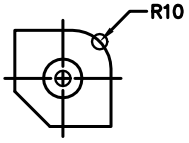
The definition points for each type of dimension are indicated in the following illustrations. The middle point of the dimension text is a definition point for all dimension types.



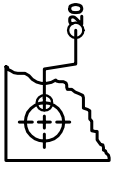

Linear: extension line origins and intersection of first extension line and dimension line



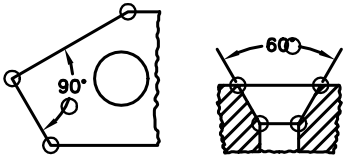

Diameter: selection point and opposite point



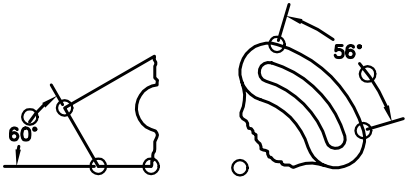

Radius: selection point and center




ordinate: feature location and leader endpoint




Three-Point Angular: angle vertex, extension line origins, and dimension line arc




Two-Point Angular: extension line origins, and dimension line arc

If no angle vertex is shown, definition points are placed at the ends of the lines that form the angle. In the two-line angular example, a definition point is placed at the center point of the dimensioned arc.

**NOTE** Definition points are drawn on a special layer named DEFPOINTS, which is not plotted.

### Modify Exploded Dimensions

You can edit exploded dimensions as you would any other objects because an exploded dimension is a collection of separate objects: lines, 2D solids, and text. Occasionally you may need to explode a dimension to make changes

such as creating a break in a dimension line or extension line. Once a dimension is exploded, you cannot reassociate the dimension into a dimension object.

**See also:**

"Control Dimension Geometry" on page 783

# Change Dimension Associativity

You may need to change the associativity of dimensions in several circumstances including adding associativity to dimensions created in previous releases.

You may need to change the associativity of dimensions in several circumstances such as the following:

- Redefine the associativity of dimensions in drawings that have been edited significantly.
- Add associativity to dimensions that have been partially disassociated.
- Add associativity to dimensions in legacy drawings.
- Remove associativity from dimensions in drawings that will be used by people working in releases prior to AutoCAD 2002, but who do not want any proxy objects in the drawings.

### Reassociate Dimensions to Different Objects

With DIMREASSOCIATE, you can select one or more dimensions and step through the extension-line origin points of each dimension. For each extension-line origin point, you can specify a new *association point* on a geometric object. Association points determine the attachment of extension lines to locations on geometric objects.

---

**NOTE** When you create or modify associative dimensions, it is important to locate their association points carefully so that if you make a future design change, the geometric objects that you change will also change the dimensions associated with them.

---

When you use the DIMREASSOCIATE command, a marker is displayed that indicates whether each successive extension line origin point of the dimension is associative or nonassociative. A square with an X in it means that the point is associated with a location on an object, while an X without the square means that the point is not associated with an object. Use an object snap to specify the new association for the extension-line origin point or press ENTER to skip to the next extension-line origin point.

**NOTE** The marker disappears if you pan or zoom with a wheel mouse.

### Change Nonassociative Dimensions to Associative

You can change all the nonassociative dimensions in a drawing to associative. Use  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 778
"Save Drawings to Previous Drawing File Formats"

### To associate or reassociate a dimension

1. Click Dimension menu ➤ Reassociate Dimensions.
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. On the command line, enter  .
2. Select one or more dimensions to disassociate and press  ENTER  when you finish.

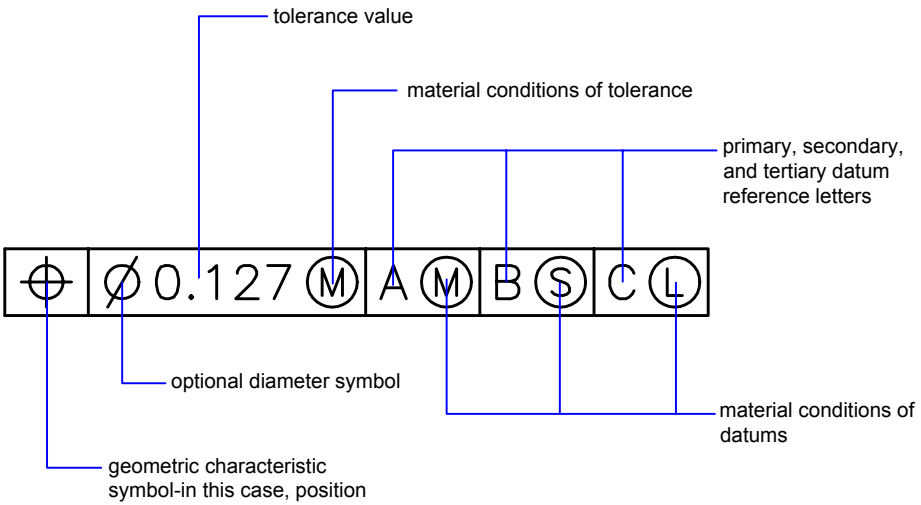
# Add Geometric Tolerances

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  or .

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.

**To create geometric tolerances**

1. Click Dimension menu ➤ 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.

Dimension toolbar

**To create a geometric tolerance with a leader**

1. On the command line, 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.

## 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.

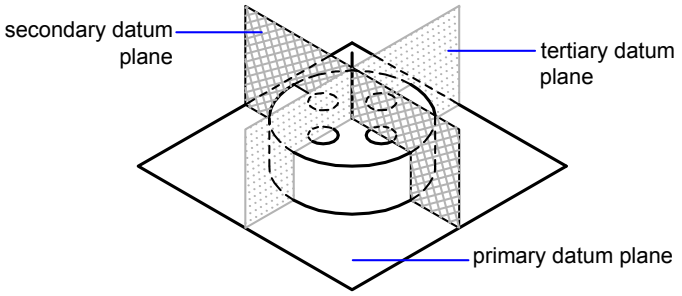
Material conditions apply to features that can vary in size:

- 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.

## 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.



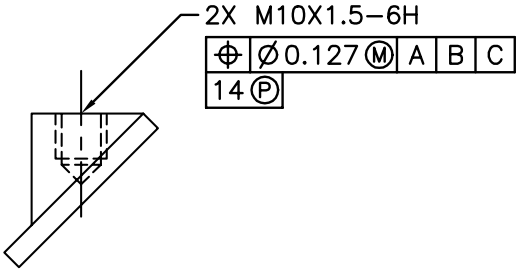

# Projected Tolerance Zones

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.

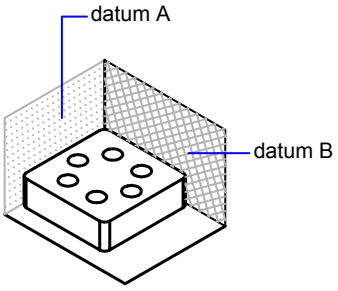



# Composite Tolerances

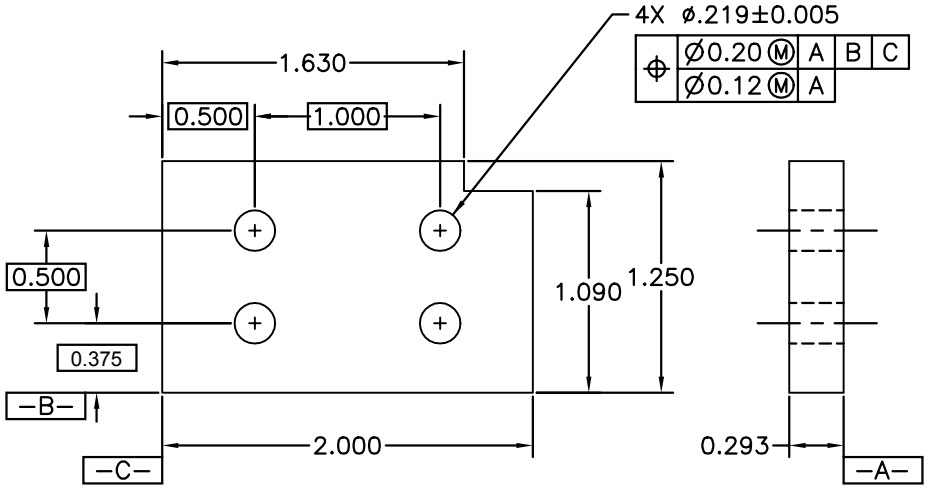
A composite tolerance specifies two tolerances for the same geometric characteristic of a feature or for features that have different datum requirements. One tolerance relates to a pattern of features and the other

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.

# Part 7
# Plot and Publish Drawings

Chapter 25 Prepare Drawings for Plotting and Publishing

Chapter 26 Plot Drawings

Chapter 27 Publish Drawings

# Prepare Drawings for Plotting and Publishing

# 26

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.

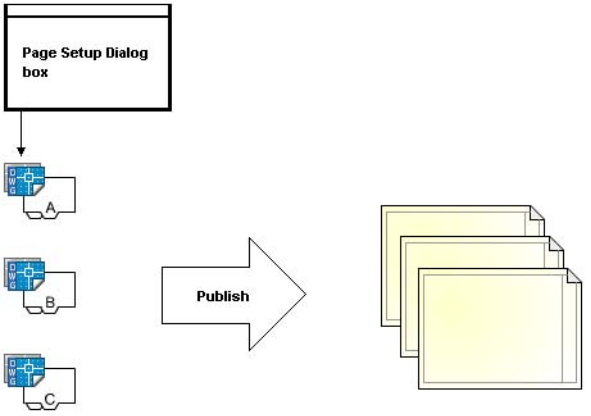
## In this chapter


- Quick Start to Preparing Drawings for Plotting and Publishing
- Specify Page Setup Settings
- Use the Layout Wizard to Specify Layout Settings
- Import PCP or PC2 Settings into a Layout
- Create and Use Named Page Setups
- Use Named Page Setups with Sheet Sets

# Quick Start to Preparing Drawings for Plotting and Publishing

Preparing a drawing for plotting or publishing requires specifying many settings and options that define the output of your drawing. These settings can be saved as a *named page setup*.

You can apply a named page setup to multiple 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.




If you want to specify the same output options for several layouts or sheets, use named page setups.

# 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.

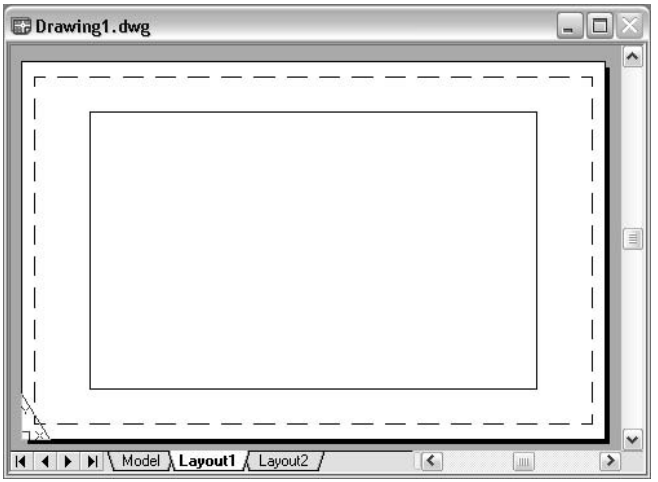
## Overview of Page Setup Settings

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. A layout is initialized if its paper size is defined in the page setup to any size other than 0 x 0. 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 File menu ➤ 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.

Layouts toolbar

Right-click a layout tab. Click Page Setup Manager.

**To apply a layout's named page setup to another layout**

1 Click the layout tab to which you want to apply another layout's page setup settings.

2 Click File menu ➤ 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.

Layouts toolbar

Right-click a layout tab. Click Page Setup Manager.

# Select a Printer or Plotter for a Layout

When you create a layout, you must select a printing or plotting device in the Page Setup dialog box in order for the layout to be printed or plotted. Once you've selected a device, you can view details about the name and location of the device, and you can change the device's configuration.

The printer or plotter you select in the Page Setup dialog box determines the printable area of the layout. This printable area is indicated by the dashed line in the layout. If you change the paper size or the printing or plotting device, it may change the printable area of your drawing page.

**See also:**

"Select a Printer or Plotter" on page 875

**"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 File menu ➤ 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.

Layouts toolbar

Right-click a layout tab and select Page Setup Manager.

### To change the configuration of a printer or plotter specified in a page setup

1. Click the layout tab for which you want to specify a printer or plotter.
2. Click File menu ➤ 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.

Layouts toolbar

Right-click a layout tab. Click Page Setup Manager.

# Select a Paper Size for a Layout

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 File menu ➤ 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.

Layouts toolbar

Right-click a layout tab. Click Page Setup Manager.

### To start the Plotter Configuration Editor

1. Click File menu ➤ 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 File menu ➤ 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 File menu ➤ Plotter Manager.
2. In the Plotter Manager, double-click the plotter configuration (PC3) file you want to edit.
3. In the Plotter Configuration Editor, Device and Document Settings tab, double-click User-Defined Paper Sizes & Calibration to display the calibration and paper size settings.
4. Select Custom Paper Sizes.
5. Under Custom Paper Sizes, click Add.
6. In the Custom Paper Size wizard, Begin page, select Use Existing.
7. In the list of existing standard paper sizes, select a paper size on which to base the custom paper size you are creating.
8. Follow the instructions in "To add a custom paper size from scratch " on page 849 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 File menu ➤ 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 File menu ➤ 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 File menu ➤ 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.

# Set the Plot Area of a Layout

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 876

**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 File menu ➤ 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.

Layouts toolbar

Right-click a layout tab. Click Page Setup Manager.

# Adjust the Plot Offset of a Layout

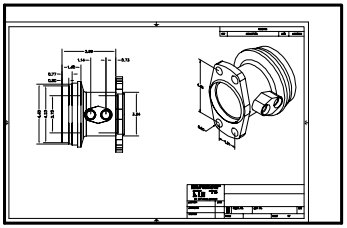
The printable area of a drawing sheet is defined by the selected output device and is represented by the dashed line in a layout. When you change the output device, the printable area may change.

The plot offset specifies an offset of the plot area relative to the lower-left corner (the origin) of the printable area or the edge of the paper, depending

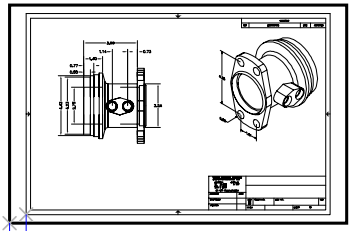
on the Specify Plot Offset Relative To option specified in the Options dialog box, Plot and Publish tab. The Plot Offset area of the Plot dialog box displays the specified plot offset option in parentheses.

You can offset the geometry on the paper by entering a positive or negative value in the X and Y Offset boxes. However, this may result in the plot area being clipped.

If you choose to plot an area other than the entire layout, you can also center the plot on the sheet of paper.



plot with origin 0,0




plot with origin -1.0,-0.5

**To adjust the plot offset of a layout**

1. Click the layout tab for which you want to adjust the plot offset.
2. Click File menu ➤ 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.

Layouts toolbar

Right-click a layout tab. Click Page Setup Manager.

# 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 .

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 216
"Draw, Scale, and Annotate in Model Space" on page 198

**To set the plot scale in a layout**

1. Click the layout tab for which you want to set the plot scale.
2. Click File menu ➤ 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.

Layouts toolbar

Right-click a layout tab and choose Page Setup.

**To set the Fit to Paper option when plotting**

1. Click the layout tab for which you want to set the plot scale to Fit to Paper.
2. Click File menu ➤ 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.

Layouts toolbar

Right-click a layout tab. Click Page Setup Manager

# Set the Lineweight Scale for a Layout

You can scale lineweights proportionately in a layout with the plot scale.

Typically, lineweights specify the line width of plotted objects and are plotted with the line width size regardless of the plot scale. Most often, you use the default plot scale of 1:1 when plotting a layout. However, if you want to plot an E-size layout that is scaled to fit on an A-size sheet of paper, for example, you can specify lineweights to be scaled in proportion to the new plot scale.

**See also:**

"Control Lineweights" on page 301

**To scale lineweights in a layout**

1. Click the layout tab for which you want to scale lineweights.
2. Click File menu ➤ 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.

Layouts toolbar

Right-click a layout tab. Click Page Setup Manager

# Select a Plot Style Table for a Layout

A plot style table is a collection of plot styles assigned to a layout or the Model tab. A plot style is an object property, similar to linetype and color. A plot style can be assigned to an object or assigned to a layer. A plot style controls an object's plotted properties.

You can also create a new plot style table to save in the page setup for the layout or edit an existing plot style table.

If you select the Display Plot Styles option under Plot Style Table (Pen Assignments), the properties of the plot styles assigned to objects are displayed in the selected layout.

**See also:**

"Control How Objects Are Plotted" on page 881

**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 File menu ➤ 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.

Layouts toolbar

Right-click a layout tab. Click Page Setup Manager

**To create a new plot style table for a layout**

1 Click the layout tab for which you want to create a new plot style table.

2 Click File menu ➤ Page Setup Manager.

3 In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.

4 Click Modify.

5 In the Page Setup dialog box, Under Plot Style Table, select New from the list.

6 Follow the instructions in the wizard that is shown (the Add Color-Dependent Plot Style Table wizard or the Add Named Plot Style Table wizard).

7 In the Page Setup dialog box, click OK.

8 In the Page Setup Manager, click Close.

Layouts toolbar

Right-click a layout tab. Click Page Setup Manager

**To edit a plot style table for a layout**

1 Click the layout tab for which you want to edit a plot style table.

2 Click File menu ➤ 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.

Layouts toolbar

Right-click a layout tab. Click Page Setup Manager

**To display plot styles in a layout**

1. Click the layout tab for which you want to display plot styles.
2. Click File menu ➤ 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.

Layouts toolbar

Right-click a layout tab. Click Page Setup Manager

# Set Shaded Viewport and Plot Options for a Layout

Shaded viewport and plot options affect how objects are plotted. The options for shaded viewport plotting give you a large degree of flexibility in conveying your three-dimensional designs to others. You can convey your design intent by choosing how viewports are plotted and by specifying resolution levels.

### Shaded Viewport Plotting Options

With shaded plotting options, you can choose whether to plot a set of shaded objects using the As Displayed, Wireframe, Hidden, or Rendered option.

Shaded viewport plotting options apply to all objects in viewports and model space. If you use the Shaded or Rendered options plot style tables included in the page setup do not affect plots. If you use the Render option, two-dimensional wireframe objects, such as lines, arcs, and text, are not plotted.

**NOTE** Shaded viewport plotting requires a raster-capable device. Most modern plotters and printers are raster-capable devices.

### Plot Options

The following options that can be specified for layouts affect how objects are plotted.

- **Plot Object Lineweights.** Specifies that lineweights assigned to objects and layers are plotted.
- **Plot with Plot Styles.** Specifies that the drawing is plotted using plot styles. Selecting this option automatically plots lineweights. If you do not select this option, objects are plotted with their assigned properties and not with the plot style overrides.
- **Plot Paper Space Last.** Specifies that objects in model space are plotted before those in paper space.
- **Hide Paperspace Objects.** Specifies whether the Hide operation applies to objects in the paper space viewport. This option is available only from a layout tab. The effect of this setting is reflected in the plot preview, but not in the layout.

**See also:**

"Set Shaded Viewport Options" on page 883
"Set Options for Plotted Objects" on page 886

### To set shaded viewport options for a layout

1. Click the layout tab for which you want to set shaded viewport options.
2. Click File menu ➤ 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.

Layouts toolbar

Right-click a layout tab and click Page Setup Manager

**To set plot options for a layout**

1 Click the layout tab for which you want to set plot options.

2 Click File menu ➤ 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.

Layouts toolbar

Right-click a layout tab. Click Page Setup Manager

# 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 File menu ➤ 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.

Layouts toolbar

Right-click a layout tab. Click Page Setup Manager

# 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 by selecting the layout, clicking Page Setup Manager on the File menu, and clicking Modify in the Page Setup Manager.

**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.

# Import PCP or PC2 Settings into a 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.

# Create and Use Named Page Setups

You can save plot device and other page setup settings as named page setups that can be modified and imported into other drawings.

You can create named page setups and apply them to other layouts in your drawing. Named page setups are saved in the drawing file and can be imported into other drawing files and applied to other layouts. You can also modify the settings of named page setups. If you modify a named page setup, you can choose whether the modifications apply to the current layout or to all the layouts in the current drawing that use the named page setup.

If you want to plot the same layout more than one way, or if you want to specify the same output options for several layouts, use named page setups.

You can apply different named page setups to the same layout to achieve specific results when plotting. For example, you might create the named page setups in the following table to control scaling and paper size.

| Page setup name | Description |
|---|---|
| NoScaling | Plot at scale 1:1, E-size sheet |
| Scale 1 to 2 | Plot at scale 1:2, C-size sheet |
| Draft | Plot to the draft-quality plotter |
| Final | Plot to the high-quality plotter |

| Page setup name | Description |
| --- | --- |
| Fit-to-Paper | Fit to Paper, A-size sheet |

Once you specify a named page setup for a layout, whenever you plot or publish the layout, it is plotted or published with the settings specified in the named page setup set for the layout.

**To create a new named page setup**

1. Click File menu ➤ 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.

Layouts toolbar

Right-click a layout tab. Click Page Setup Manager

**To apply a named a page setup to a layout**

1. Click the layout tab for which you want to apply a named page setup.
2. Click File menu ➤ 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.

Layouts toolbar

Right-click a layout tab. Click Page Setup Manager

**To modify a named a page setup**

1 Click File menu ➤ 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.

Layouts toolbar

Right-click a layout tab. Click Page Setup Manager

**To import named page setups from another drawing**

1 Click File menu ➤ 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.

Layouts toolbar

Right-click a layout tab. Click Page Setup Manager

**To delete a named page setup**

1 Click File menu ➤ 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.

Layouts toolbar

Right-click a layout tab and click Page Setup Manager

**To rename a page setup**

1. Click File menu ➤ 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.

Layouts toolbar

Right-click a layout tab. Click Page Setup Manager

# Use Named Page Setups with Sheet Sets

You can use named page setups to specify the same output options for all the sheets in a sheet set.

You can create a sheet from scratch using the Sheet Set Manager, and then apply a named page setup to the layout.

You can also apply named page setups that are stored in the sheet set's page setup overrides DWT file to a single sheet or to an entire sheet set for a one-time publish operation.

**See also:**

"Create and Use Named Page Setups" on page 863

**To apply a named page setup to a sheet created from scratch**

1. Click File menu ➤ Open Sheet Set.
2. In the Open Sheet Set dialog box, select a sheet set. Click Open.
3. In the Sheet Set Manager, right click in the Sheets area. Click New Sheet.
4. In the New Sheet dialog box, enter a number and sheet title. Click OK.
5. In the Sheet Set Manager, under Sheets, double-click the new sheet.
6. Click File menu ➤ Page Setup Manager.
7. In the Page Setup Manager, under Page Setups, select a named page setup. Click Set Current.

   Named page setups are not enclosed in asterisks.
8. Click Close.

Standard toolbar

**To apply a named page setup as an override when publishing sheets**

1. Click File menu ➤ Open Sheet Set.
2. In the Open Sheet Set dialog box, select a sheet set. Click Open.
3. In the Sheet Set Manager, in the Sheets area, select a sheet set, a subset, or a sheet to publish.
4. At the top of the Sheet Set Manager, click the Publish button.
5. On the shortcut menu, place your cursor over Publish Using Page Setup Override.
6. On the submenu, click the named page setup you want to use.

Standard toolbar

# Plot Drawings

# 27

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.

## In this chapter

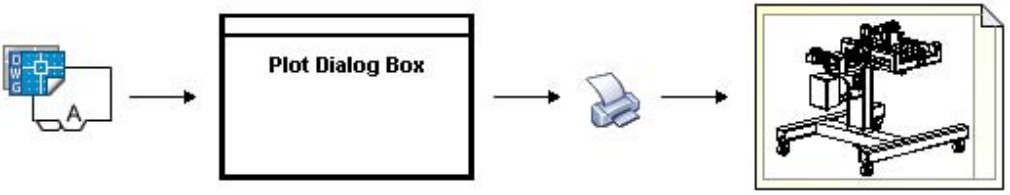

- Quick Start to Plotting
- Overview of Plotting
- Use a Page Setup to Specify Plot Settings
- Select a Printer or Plotter
- Specify the Area to Plot
- Set Paper Size
- Position the Drawing on the Paper
- Control How Objects Are Plotted
- Preview a Plot
- Plot Files to Other Formats

# Quick Start to Plotting

To print a single layout or part of a drawing, use the Plot dialog box.

Use a named page setup or change the settings in the Plot dialog box to define the output of your drawing.

To output more than one drawing, use the Publish dialog box.




**See also:**

"Quick Start to Publishing" on page 920

# 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 nonsystem 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.

### 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.

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**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

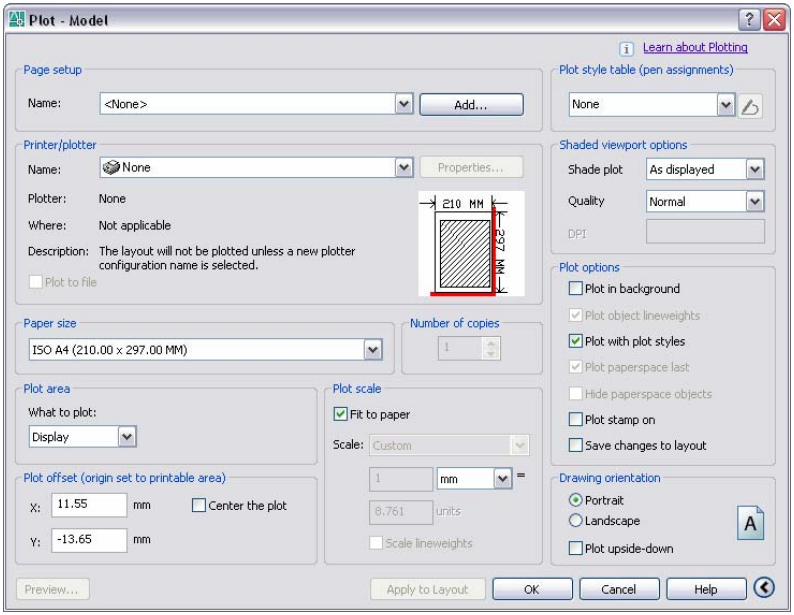
version and used in a commercial version will also result in the educational plot stamp being plotted.

**See also:**

"Create Multiple-View Drawing Layouts (Paper Space)" on page 203
"To view the custom properties Help" in the Driver and Peripheral Guide

### To plot a drawing

1 Click File menu ➤ 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.

Standard toolbar

Right-click the Model tab or a layout tab and click Plot.

To plot on the command line instead of in a dialog box, use -PLOT.

**To turn background plotting on or off**

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Plot and Publish tab, under Background Processing Options, select or clear the Enable Background Plot When Plotting option.
3. Click OK.

   **NOTE** When you plot in the background, you can return immediately to your drawing. While a plot job is being processed in the background, you can check the status of your job by placing your cursor over the plotter icon in the status tray. You can also view details about all completed jobs from the current program session.

**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.

Status tray

**To cancel part or all of a plot job that is processing in the background**

1. In the status tray, right-click the plotter icon. Click Cancel Sheet <*sheetname*> or Cancel Entire Job.

Status tray

**To view details about jobs you have plotted**

1. Do one of the following:
   - Click File menu ➤ View Plot and Publish Details.
   - In the status tray, click the plotter icon.
2. In the Plot and Publish Details dialog box, view details about plotted jobs.

In the status tray, right-click the plotter icon. Click View Plot and Publish Details.

# Use a Page Setup to Specify Plot Settings

You can use a page setup to specify the settings for your plot job. When you select a page setup in the Plot dialog box, the settings from the page setup are added to the Plot dialog box. You can choose to plot with those settings, or change any of the settings individually and then plot.

Any settings you specify in the Plot dialog box can be saved as a new named page setup by clicking the Add button in the Page Setup area.

Any settings specified in the Plot dialog box, whether you've applied a page setup from the Page Setup list, or changed the settings individually, can be saved to the layout for use the next time you plot.

**To plot a drawing using the settings specified in a page setup**

1. Click File menu ➤ 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.

Standard toolbar

**To save plot settings as a new named page setup**

1 Click File menu ➤ 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.

Standard toolbar

**To save plot settings to the layout**

1 With a layout tab selected, click File menu ➤ Plot.

2 In the Plot dialog box, under Page Setups, select a page setup or specify settings individually.

3 Click Apply to Layout.

Standard toolbar

# Select a Printer or Plotter

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 File menu ➤ 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.

Standard toolbar

# 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  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 File menu ➤ 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.

Standard toolbar

# 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 nonsystem 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 nonsystem 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 File menu ➤ 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.

Standard toolbar

**To set the default paper size for a layout**

1 Click File menu ➤ Page Setup Manager.

2 In Page Setup Manager, Page Setups area, the layout for which you want to set the paper size should be selected. If not, select the layout. Click Modify.

3 In the Page Setup dialog box, under Paper Size, select a paper size from the list. Click OK.

The paper sizes that are listed depend on the plotter specified in the page setup.

4 In Page Setup Manager, click Close.

The layout reflects the changes.

Layouts toolbar

**To set the default paper size for a plotter**

1 Click File menu ➤ 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 nonsystem 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.

Layouts toolbar

**To create or edit a custom paper size for a nonsystem printer**

1 Click File menu ➤ Plotter Manager.

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 nonsystem 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.

# 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.

# 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. 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.

# 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.

# 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.

---

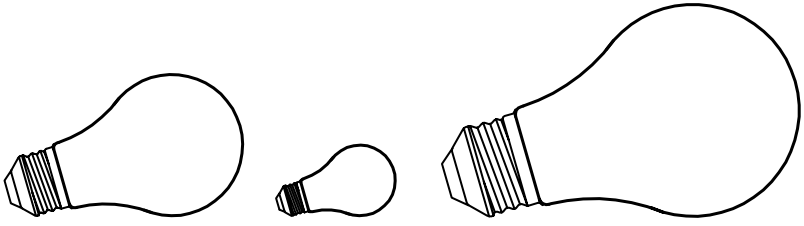
**NOTE** You can modify the list of scales that are displayed in all view and plot scale lists with .

---

#### Set a Specific Scale

When you plot, the paper size you select determines the unit type, inches or millimeters. For example, if the paper size is in mm, entering **1** under mm and **10** under Units produces a plotted drawing in which each plotted millimeter represents 10 actual millimeters.

The illustrations show a light bulb plotted at three different scales.




### Scale the Drawing to Fit the Page

When you review drafts, a precise scale is not always important. You can use the Fit to Paper option to plot the view at the largest possible size that fits the paper. The height or width of the drawing is fit to the corresponding height or width of the paper.

When you plot a perspective view from model space, the view is scaled to fit the paper even when you enter a scale.

When you select the Fit to Paper option, the text boxes change to reflect the ratio of plotted units to drawing units. This scale is updated whenever you change the paper size, plotter, plot origin, orientation, or size of the plotted area in the Plot dialog box.

---

**NOTE** This option is not available when the Plot Area is set to Layout.

---

#### To plot using a real-world scale

1. Click File menu ➤ Plot.
2. In the Plot dialog box, under Plot Scale, select a scale from the Scale box.
3. Click OK to plot the drawing.

Standard toolbar

#### To plot using a custom scale

1. Click File menu ➤ 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.

Standard toolbar

**To scale a drawing to fit the page**

1 Click File menu ➤ 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.

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**NOTE** This option is not available when the Plot Area is set to Layout.

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Standard toolbar

# Set Shaded Viewport Options

You can choose among several options for plotting shaded and rendered viewports. You can plot a viewport as it is displayed, in wireframe, with hidden lines removed, or as rendered.

You can specify how each viewport should be plotted, and save the plot settings with a drawing. You also can choose from a wide variety of resolutions, up to the resolution of your plotter, and save the resolution settings with a drawing.

## Overview of Shaded Viewport Plotting

The options for shaded viewport plotting give you a large degree of flexibility in conveying your three-dimensional designs to others. You can convey your design intent by choosing how viewports are plotted and by specifying resolution levels.

With shaded plotting options, you can choose whether to plot a set of shaded objects using the As Displayed, Wireframe, Hidden, or Rendered option. Shaded and rendered viewports are plot-previewed, plotted, plotted to file, and published with full shading and rendering.

You can use realistic plots in your presentations by plotting viewports as they are displayed on the screen or otherwise.

Shaded viewport plotting options apply to all objects in viewports and model space. If you use the Shaded or Rendered options plot style tables included in the page setup do not affect plots. If you use the Render option, two-dimensional wireframe objects, such as lines, arcs, and text, are not plotted.

**NOTE** Shaded viewport plotting requires a raster-capable device. Most modern plotters and printers are raster-capable devices.

## Specify Shaded Plotting Settings

If you are plotting a drawing that contains 3D solids that are shaded, you can control how the drawing is plotted.

Specifically, you can choose from the following options:

- **As Displayed.** Plots the design as it is displayed; all the shading is preserved.
- **Wireframe.** Displays lines and curves to represent object boundaries.
- **Hidden.** Suppresses the plotting of objects that are located behind other objects.
- **Rendered.** Renders objects before they are plotted, based on Render options you set before you plot.

You can select an option for your drawing either from model space or from a layout. From model space, the options are available in the Properties palette and the Plot dialog box. From a layout, after you select a viewport, the options are available from the shortcut menu and from the Properties palette.

Either way, you can save your settings with the drawing.

**NOTE** If you select the Rendered option, specify Render settings before plotting. If the Rendered option is used for a highly complex set of objects, the hardcopy output might contain only the viewport border.

**To modify the shade plot setting of a viewport**

1. Make sure you are on a layout tab.
2. Double-click the border of the viewport you want to modify.
3. In the Properties palette, under Misc., select Shade Plot, and then select an option for plotting.

Select the viewport and right-click in the drawing area. Click a setting from the Shade Plot options.

**To modify the shade plot setting in model space**

1. Make sure you are on the Model tab and no objects are selected.
2. Click File menu ➤ Plot.
3. In the Plot dialog box, under Shaded Viewport Options, Shade Plot drop-down list, select an option.

## Specify a Resolution Level for Shaded Plotting

You can set the resolution of shaded plots for either greater speed or higher fidelity.

After you select an appropriate plotter, you can specify the level of quality for plotted output. The quality level determines the dpi. The dpi that corresponds to a quality level is based on the plotter you select.

The maximum dpi available is also based on the plotter you select. You can specify a custom quality level and directly change the dpi to a setting between 100 and the maximum dpi of the plotter.

The higher the fidelity, the more computer memory is used, so the longer it takes to plot. High fidelity is not necessary for all plots, and a setting between 300 and 600 dpi is generally sufficient for most plots.

**To specify a resolution level for shaded plotting**

1. Click File menu ➤ Plot.
2. In the Plot dialog box, under Shaded Viewport Options, Quality drop-down list, select an option.

If you want to enter a custom dpi, select Custom and specify a dpi in the DPI box.

3 Click OK.

# Set Options for Plotted Objects

In the Plot and the Page Setup dialog boxes, you can choose from options that affect how objects are plotted.

- **Shaded Viewport Plotting.** Specifies shaded plotting options: As Displayed, Wireframe, or Hidden. The effect of this setting is reflected in the plot preview, but not in the layout.
- **Plot Object Lineweights.** Specifies that lineweights assigned to objects and layers are plotted.
- **Plot with Plot Styles.** Specifies that the drawing is plotted using plot styles. Selecting this option automatically plots lineweights. If you do not select this option, objects are plotted with their assigned properties and not with the plot style overrides.
- **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 File menu ➤ Plot.

2 In the Plot dialog box, Shaded Viewport Options area, Shade Plot drop-down list, select an option.

Standard toolbar

**To set shaded plotting options from a layout tab**

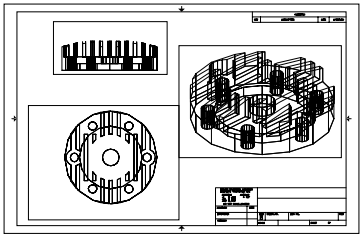
1. Select a layout viewport.
2. Double-click the viewport border to display the Properties palette.
3. On the Properties palette, click Shade Plot.
4. In the Shade Plot drop-down list, select an option.

Standard toolbar

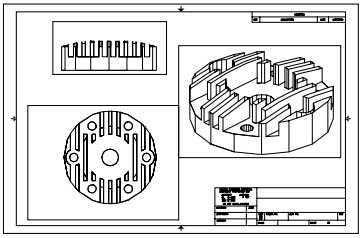
Select the viewport, right-click in the drawing area, and then click Shade Plot.

**To remove hidden lines when plotting from the Model tab**

1. Click File menu ➤ Plot.
2. In the Plot dialog box, under Shaded Viewport Options area, Shade Plot drop-down list, select Hidden.
3. Click OK to plot the drawing.



Hidden lines not removed

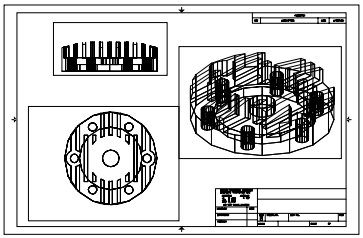


Hidden lines removed

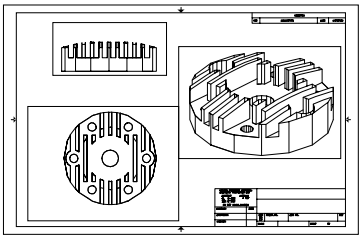
Standard toolbar

**To remove hidden lines when plotting from a layout tab**

1. Select a layout viewport.
2. Double-click the viewport border to display the Properties palette.
3. On the Properties palette, click Shade Plot.
4. In the Shade Plot drop-down list, select Hidden.
5. Click OK to plot the drawing.



Hidden lines not removed



Hidden lines removed

Standard toolbar

Select the viewport and right-click in the drawing area. Click Shade Plot.

**To plot lineweights**

1. Click File menu ➤ 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.

Standard toolbar

**To turn off plot styles**

1. Click File menu ➤ Plot.
2. In the Plot dialog box, under Plot Options, clear the Plot with Plot Styles option.
3. Click OK to plot the drawing.

Standard toolbar

**To change the order in which objects are plotted**

1. Click File menu ➤ 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.

Standard toolbar

**To hide paper space objects when plotting from a layout tab**

1. Click File menu ➤ Plot.
2. In the Plot dialog box, under Plot Options, select Hide Paperspace Objects.
3. Click OK to plot the drawing.

Standard toolbar

# Use Plot Styles to Control Plotted Objects

You can control many aspects of how an object is plotted by using plot styles.

## Overview of Plot Styles

A plot style is an object property, similar to linetype and color. A plot style can be assigned to an object or assigned to a layer. A plot style controls an object's plotted properties, including

- Color
- Dither
- Grayscale
- Pen number
- Virtual pen
- Screening
- Linetype
- Lineweight
- 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.

## 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.

## 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  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  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** on the command line 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.

## 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.

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**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 File menu ➤ 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** on the command line to display the plot style settings.

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Layouts toolbar

**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 File menu ➤ Page Setup.
3. In Page Setup Manager, click Modify.
4. In the Page Setup dialog box, under Plot Style Table (Pen Assignments), select the Display Plot Styles option.
5. Click OK.
6. In Page Setup Manager, click Close.

   The effects of the plot style table are previewed in the layout.

Layouts toolbar

## Manage Plot Style Tables

You can use the Plot Style Manager to add, delete, rename, copy, and edit plot style tables.

Color-dependent (CTB) and named (STB) plot style tables are stored in the *Plot Styles* folder by default. This folder is also known as the Plot Style Manager.

You can use the Plot Style Manager to add, delete, rename, copy, and edit plot style tables. The Plot Style Manager lists all of the available plot style tables.

**To create a plot style table**

1. Click Tools menu ➤ Wizards ➤ Add Plot Style Table.
2. Read the first page. Click Next.
3. On the Begin page, you can choose to use an configuration file (CFG) or plotter configuration file (PCP or PC2) to import pen settings, base the new plot style table on an existing plot style table, or start from scratch. If you use an existing plot style table, the new plot style table uses the same type of table as the original. Click Next.
4. On the Pick Plot Style Table page, select Color-Dependent Plot Style Table or Named Plot Style Table.
5. If you are importing pen settings from a PCP, PC2, or CFG file, or if you are basing the new plot style table on an existing plot style table, specify

the file on the Browse File Name page. If you use a CFG file, you may need to select which plotter's configuration to import. Click Next.

6. On the File Name page, enter a name for the new plot style table. Click Next.
7. On the Finish page, you can edit the new plot style table by choosing Plot Style Table Editor. You can assign the new plot style table so that it can be used in all drawings.
8. Click Finish.

   The new plot style table is available in both the Plot and the Page Setup dialog boxes for all drawings using color-dependent plot style tables.

#### To rename a plot style table

1. Click File menu ➤ 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 File menu ➤ 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 File Menu ➤ 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.

# Use Color-Dependent Plot Style Tables

When a drawing uses color-dependent plot style tables, you cannot assign a plot style to individual objects or layers. Instead, to assign plot style properties to an object, you change the color of the object or layer.

You can assign color-dependent plot style tables to layouts. You can use several predefined color-dependent plot style tables, edit existing plot style tables, or create your own.

Color-dependent plot style tables are stored in the *Plot Styles* folder and have a *.ctb* extension.

### Use Predefined Color-Dependent Plot Style Tables

Several color-dependent plot style tables are installed in the *Plot Styles* folder, also known as the Plot Style Manager.

| Table | Description |
|---|---|
| *acad.ctb* | Default plot style table |
| *fillPatterns.ctb* | Sets first 9 colors to use first 9 fill patterns, all others to use object's fill |
| *grayscale.ctb* | Converts all colors to grayscale when plotted |
| *monochrome.ctb* | Plots all colors as black |
| None | Applies no plot style table |
| *screening 100%.ctb* | Uses 100% ink for all colors |
| *screening 75%.ctb* | Uses 75% ink for all colors |
| *screening 50%.ctb* | Uses 50% ink for all colors |
| *screening 25%.ctb* | Uses 25% ink for all colors |

**NOTE** You can assign a color-dependent plot style table to a layout only if the drawing has been set to use color-dependent plot style tables.

**See also:**

"Assign Plot Style Tables to Layouts" on page 893

# 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. Right-click in the drawing area. 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.

Standard toolbar

**To change a layer's plot style**

1. Click Format menu ➤ Layer.
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.

Layers toolbar

**To set the current plot style**

1. Click Format menu ➤ Plotstyle.
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.

## 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 902

### To create a named plot style

1 Click File menu ➤ Plot Style Manager.

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.

### To copy a named plot style

1 Click File menu ➤ Plot Style Manager.

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.

#### To change a plot style's description

1. Click File menu ➤ Plot Style Manager.
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.

#### To rename a named plot style

1. Click File menu ➤ Plot Style Manager.
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.

#### To delete a named plot style

1. Click File menu ➤ Plot Style Manager.
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.

## Use Predefined Named Plot Style Tables

One additional named plot style table is installed for you to use beyond the default plot style table. All named plot style tables have an *.stb* extension.

- *acad.stb*: Default plot style table
- *Monochrome.stb*: All colors plot as black
- None: No plot style table applied

---

**NOTE** Named plot style tables are available only if the drawing has been set to use named plot style tables.

---

**See also:**

"Assign Plot Style Tables to Layouts" on page 893

## 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 File menu ➤ 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.

# 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 File menu ➤ 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.

# 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.

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.

**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 File menu ➤ 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 File menu ➤ 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 File menu ➤ 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 File menu ➤ 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.

## 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 File menu ➤ 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.

## 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 293
"Control Lineweights" on page 301

### To set the plotted lineweight

1. Click File menu ➤ 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 File menu ➤ 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 File menu ➤ 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.

## Assign Plotted Line End and Join Styles

You can set the line end and join styles for objects that have lineweight assigned, either as an object property or as a plot style override.

### Assign Line End Style

The program includes the following line end style options:

- Butt
- Square
- Round
- Diamond

The default setting for Line End Style is Use Object End Style, which is rounded. Assign a line end style in a plot style to override the object's default line end style at plot time.

**NOTE** SHX text plots best with the Round End and Round Join styles.

### Assign Line Join Style

The program includes the following line join style options:

- Miter
- Bevel
- Round
- Diamond

The default setting for Line Join Style is Use Object Join Style, which is rounded. Assign a line join style in a plot style to override the object's default line join style at plot time.

**To assign a line end or line join style**

1. Click File menu ➤ 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.

## Assign Plotted Fill Styles

The program includes the following fill style options when plotting wide polylines, donuts, objects hatched with a solid fill, and solids:

- Solid
- Checkerboard
- Crosshatch
- Diamonds
- Horizontal Bars
- Slant Left
- Slant Right
- Square
- Dots
- Vertical Bar

The default setting for Fill Style is Use Object Fill Style. Assign a fill style in a plot style to override the object's fill style at plot time.

You can apply a global scale factor to non-ISO linetypes and fill patterns in plot styles.

**See also:**

"Choose Hatch Patterns and Solid Fills" on page 686

**To assign a fill style**

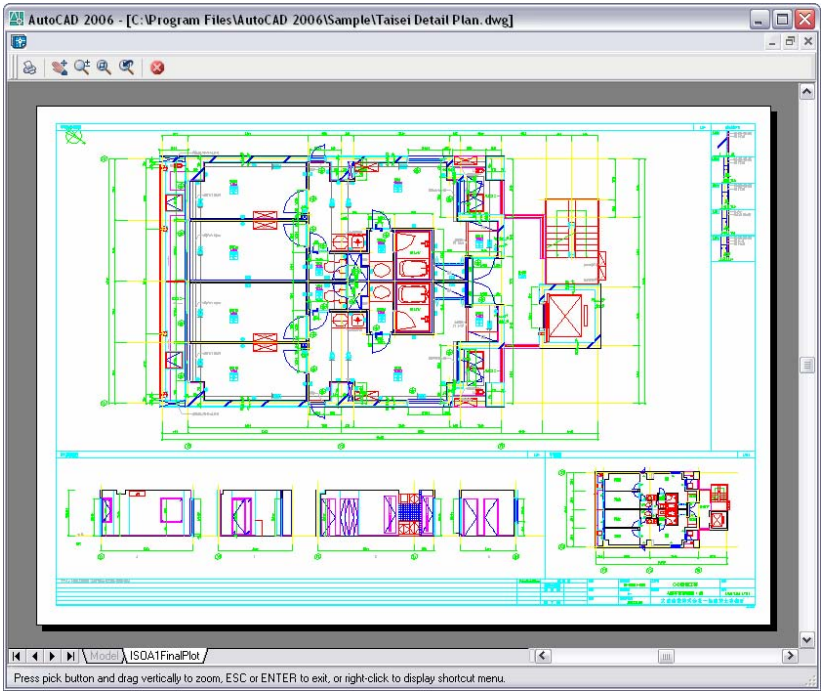
1. Click File menu ➤ Plot Style Manager.
2. Right-click a CTB or STB file. Click Open.
3. In the Plot Style Table Editor, Table View tab, click the Fill Style field for the plot style you want to change.
4. Select a fill style from the drop-down list.

# 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 File menu ➤ 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.

Standard toolbar

# Plot Files to Other Formats

You can export or plot your drawings in a number of formats, including DWF, DXF, and Windows metafile (WMF). You can also output your drawings in image formats using specially designed plotter drivers.

In each case, a nonsystem plotter driver is configured to output file information. You can control the custom properties of each nonsystem driver in the Plotter Configuration Editor. Specific help for each driver is also available by choosing Help while in the custom Properties dialog box for the individual driver (accessed through the Plotter Configuration Editor).

**See also:**

"Publish Drawings" on page 919

# Plot DWF Files

You can use the program to create Design Web Format (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® DWF™ Composer or Autodesk® DWF™ Viewer. With Autodesk DWF Composer or Autodesk DWF 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 919

### To plot a DWF file

1. Click File menu ➤ 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.

Standard toolbar

# Plot to DXB File Formats

DXB (drawing interchange binary) file formats are supported using the DXB nonsystem file driver. This is commonly used to "flatten" 3D drawings to 2D.

The output is compatible with the DXBIN command and with the ADI DXB driver delivered with earlier releases. The DXB driver shares these limitations of the ADI driver:

- The driver produces 16-bit integer DXB files containing only vectors.
- DXB output is monochrome; all vectors are color 7.
- Raster images and embedded OLE objects are not supported.
- The driver ignores object and plot style lineweights.

**See also:**

**"Configure for File Output" in the Driver and Peripheral Guide**

**To create a DXB file**

1 Make sure you have configured a plotter driver for DXB file output. (See Configure for File Output in the *Driver and Peripheral Guide*.)

2 Click File menu ➤ 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.

Standard toolbar

# 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 File menu ➤ 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.

Standard toolbar

# 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 nonsystem 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:**

**"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 File menu ➤ 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.

Standard toolbar

# 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.

For example, HP-GL and HP-GL/2 formats are used as an intermediate format for illustration and manufacturing applications, for archive purposes, and for use with a wide range of output devices.

The HP-GL nonsystem driver supports HP-GL (Hewlett-Packard Graphics Language), a widely used pen plotter language that has vector-only capabilities. Raster objects are not supported by the HP-GL device driver.

The nonsystem HP-GL/2 driver supports a variety of HP-GL/2 pen plotters and ink-jet plotters. It is a generic HP-GL/2 driver that is not optimized for

any particular manufacturer's devices. For example, it does not send PJL commands to devices, as a true Hewlett-Packard driver would. The HP-GL/2 driver supports obsolete pen plotters and newer devices made by manufacturers other than Hewlett-Packard.

**See also:**

**"Set Device-Specific Configurations" in the Driver and Peripheral Guide**

**To create a plot file**

1. Click File menu ➤ Plot.
2. In the Plot dialog box, under Printer/Plotter, in the Name box, select a plotter configuration.
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.

Standard toolbar

# Publish Drawings

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Publishing provides a streamlined alternative to plotting multiple drawings. You can easily publish an entire sheet set from the Sheet Set Manager as a set of paper drawings or as a single, electronic, multisheet Design Web Format (DWF) file.

Publishing an electronic drawing set as a DWF file saves time and increases productivity by providing accurate, compressed representations of drawings in a file that's easy to distribute and view. DWF files can be viewed or plotted by anyone using a free, downloaded DWF file viewer, Autodesk® DWF™ Viewer.

## In this chapter


- Quick Start to Publishing
- Overview of Publishing
- Create and Modify a Drawing Set for Publishing
- Create a Paper or Plot File Drawing Set
- Publish an Electronic Drawing Set
- Publish a Sheet Set
- Republish a Drawing Set
- View a Published Electronic Drawing Set
- Set Publish Options
- 3D DWF Publishing
- Configure a DWF6 Driver (Advanced)

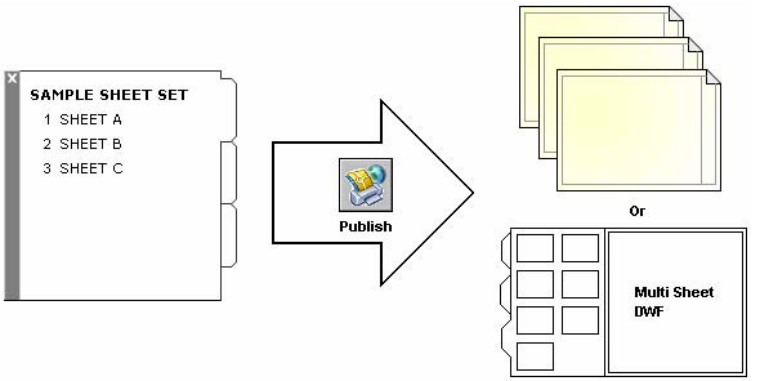
# Quick Start to Publishing

Sets of drawings are the primary deliverable for most design groups. Creating a drawing set to distribute for review can be complicated and time consuming.

Using the Publish dialog box, you can easily assemble a collection of drawings and, with one click, create a paper drawing set or an electronic drawing set.

Electronic drawing sets are saved as Design Web Format™ (DWF™ ) files. You can view or plot DWF files using a free DWF file viewer, Autodesk® DWF™ Viewer.

To output a single drawing, use the Plot dialog box.




**See also:**

"Quick Start to Plotting" on page 870

No procedures for this topic.

# Overview of Publishing

Publishing provides an easy way to create a paper drawing set or 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 Design Web Format™ (DWF™ ) file.

You can publish an entire sheet set from the Sheet Set Manager. With one click, you can create an electronic drawing set by publishing the sheet set to a single, multisheet DWF file.

You can create a paper drawing set by publishing the sheet set to the plotter named in each sheet's page setup.

Using the Publish dialog box, you can assemble a collection of drawings to publish and save the list as a Drawing Set Descriptions (DSD) file. You can customize this collection of drawings for a specific user, and you can add and remove sheets as a project evolves. Once you've created a list of drawing sheets in the Publish dialog box, you can publish the drawings to any of the following:

- The plotter named in each sheet's page setup (including drawings that you want to plot to file)
- A single, multisheet DWF file
- Multiple single-sheet DWF files

Using 3D DWF Publishing, you can create and publish DWF files of your three-dimensional models. As a technology preview in AutoCAD® 2006, the 3DDWFPUBLISH command is, by default, installed in all network installations, and an optional feature in stand-alone installations.

Publishing an electronic drawing set as a DWF file saves time and increases productivity by providing accurate, compressed representations of drawings in a file that's easy to distribute and view. This also maintains the integrity of your original drawings.

Publishing generates DWF6 files, which are created in a vector-based format (except for inserted raster image content), ensuring that precision is maintained. You can view or plot DWF files using a free DWF file viewer, Autodesk® DWF™ Viewer. DWF files can be distributed using email, FTP sites, project websites, or CDs.

You can specify which block-related properties and attributes you want to make available to others who are using the Autodesk DWF Viewer to view or print your published DWF files. For example, you can publish a DWF file for a plumbing contractor that contains block attribute information about the plumbing fixtures specified in your drawing data. And, from the same set of sheets, you can include only the block attribute data about light fixtures for an electrical contractor.

By default, jobs that are published are processed in the background, so that you can return immediately to your drawing. Only one job that you have published can be processed in the background at a time. While a job is being processed in the background, you can check its status by placing the cursor over the plotter icon on the right side of the status bar. You can also view details about all completed jobs that you have plotted or published from the current session.

**See also:**

"Publish, Transmit, and Archive Sheet Sets" on page 253

"Specify Page Setup Settings" on page 844
"3D DWF Publishing" on page 948

### 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.

Status tray

### To cancel part or all of a published job that is processing in the background

1. Right-click the plotter icon in the status tray. Click Cancel Sheet <*sheetname*> or Cancel Entire Job.

Status tray

### To view details about jobs you have published

1. Do one of the following:
   - Click File menu ➤ View Plot and Publish Details.
   - In the status tray, left-click the plotter icon.
2. In the Plot and Publish Details dialog box, view details about published jobs.

In the status tray, right-click the plotter icon. Click View Plot and Publish Details.

### To turn background publishing on or off

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Plot and Publish tab, under Background Processing Options, select or clear the Enable Background Plot When Publishing option.
3. Click OK.

# Create and Modify a Drawing Set for Publishing

You can assemble a collection of drawing sheets to publish to a plotter, plot files, or a DWF™ file. You can customize your drawing set for a specific user, and you can add, remove, reorder, copy, and rename sheets in a drawing set as a project evolves.

You can publish the drawing set directly to paper or to a single DWF file or multiple DWF files that can be distributed using email, FTP sites, project websites, or CD. You can save a description of a drawing set that's been assembled to publish in a Drawing Set Descriptions (DSD) file.

### To create a drawing set for publishing

1. With a drawing open, click File menu ➤ 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 must remove those sheets that you don't want to become a 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, make sure the Include Layouts When Adding Sheets option is selected on 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, make sure the Include Model When Adding Sheets option is selected either on 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. 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** It is important that you change the page setup for each layout depending on your desired output. Only model space page setups can be applied to model space sheets, and only paper space page setups can 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.

Standard toolbar

**To add sheets from a drawing to a drawing set for publishing**

1. Click File menu ➤ Publish.
2. In the Publish dialog box, click the Add Sheets button.
3. In the Select Drawings dialog box, select drawings. Click Select to add them to the list of sheets in the Publish dialog box.

   **NOTE** You can also drag drawings from the desktop into the Publish dialog box to add sheets to the list.

   All of the layouts in a drawing become individual sheets in the list of drawing sheets.

   You must remove the drawing sheets that you don't want to become a part of the drawing set. Layouts must be initialized before they can be published. (A layout is initialized if its paper size is defined in the page setup to any size other than 0 x 0.)

Standard toolbar

**To add sheets from a DSD file to a drawing set for publishing**

1. Click File menu ➤ Publish.
2. In the Publish dialog box, click the Load Sheet List button.
3. In the Load List of Sheets dialog box, select a DSD file. Click Load.
4. In the Replace or Append dialog box, click Replace to replace the current sheets with the sheets in the DSD file, or click Append to add the sheets in the DSD file to the list of sheets in the Publish dialog box.

Standard toolbar

**To remove a sheet from a drawing set for publishing**

1. Click File menu ➤ Publish.
2. In the Publish dialog box, make sure the sheets that you want to remove are listed.
3. Under Sheets, select one or more sheets to remove. Click the Remove Sheets button.

Standard toolbar

**To remove all sheets from a drawing set for publishing**

1. Click File menu ➤ Publish.
2. In the Publish dialog box, make sure the sheets that you want to remove are listed.
3. In the Sheet List area, right-click. Click Remove All.

**NOTE** Removing all sheets cannot be undone.

Standard toolbar

**To reorder sheets in a drawing set for publishing**

1. Click File menu ➤ Publish.
2. In the Publish dialog box, make sure the sheets that you want to reorder are listed.
3. Select a sheet. Click either the Move Sheet Up or Move Sheet Down button.

**NOTE** Sheets in the drawing set are viewed or plotted in the order shown in the Publish dialog box.

Standard toolbar

**To copy sheets in a drawing set for publishing**

1. Click File menu ➤ Publish.
2. In the Publish dialog box, make sure the sheets that you want to copy are listed.
3. Select one or more sheets in the Sheets 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.

Standard toolbar

**To rename sheets in a drawing set for publishing**

1. Click File menu ➤ Publish.
2. In the Publish dialog box, make sure the sheets that you want to rename are listed.
3. Select a sheet in the Sheet list, and then do one of the following:

   - Right-click. Click Rename Sheet.
   - Press F2.

4. Enter the new sheet name.

Standard toolbar

**To change the page setup of one sheet in a drawing set for publishing**

1. Click File menu ➤ Publish.
2. In the Publish dialog box, make sure the sheet for which you want to change the page setup is listed.
3. Select a sheet in the Sheet list.
4. Under Page Setup, select a page setup to apply to the drawing sheet.

Standard toolbar

**To change the page setup of one or more sheets in a drawing set for publishing**

1. Click File menu ➤ Publish.
2. In the Publish dialog box, make sure the sheets for which you want to change the page setups are listed.
3. Select one or more sheets in the list and right-click. Click Change Page Setup.
4. In the Page Setup list, select a page setup to apply to the drawing sheets.

**NOTE** It is important that you change the page setup for each layout depending on your desired output. Only model space page setups can be applied to model space sheets, and only paper space page setups can be applied to paper space sheets.

Standard toolbar

**To import a page setup from another drawing to apply to a drawing sheet for publishing**

1. Click File menu ➤ Publish.
2. In the Publish dialog box, make sure the sheet for which you want to change the page setup is listed.
3. Select a sheet in the Sheet list.

4 Under Page Setup, select Import from the list.

5 In the Import Page Setups dialog box, select the drawing whose page setups you want to import. Click Import.

Standard toolbar

**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.

# Create a Paper or Plot File Drawing Set

You can publish a drawing set to a plotter or plot file.

You can assemble drawing sheets into a customized drawing set and publish the sheets to the plotter named in the page setup specified for each sheet. If the plot device named in the page setup is a paper plotter, then your output will be a paper drawing set.

If the plotter is configured to plot to a file, the sheets are saved to files in the plot file location specified in the Publish Options dialog box. Each drawing sheet's plot file is saved with the same name as the sheet, with the appropriate file extension for the file (for example, *.plt*, *.jpg*, or *.bmp*). The default location can be changed in the Options dialog box, Plot and Publish tab, under Plot to File.

**See also:**

"Set Publish Options" on page 940

**To create and publish a paper or plot file drawing set**

1 With a drawing open, click File menu ➤ Publish.

2 In the Publish dialog box, the drawing layouts are shown in the Sheets List. To create the drawing set, modify the list of drawing sheets by doing any of the following as needed:

- **Add sheets.** To add sheets from other drawings, click the Add Sheets button (or drag drawings from the desktop). In the Select Drawings dialog box, select drawings. Click Select to add them to the list of sheets in the Publish dialog box. All of the layouts in a drawing become individual sheets in the list of drawing sheets. You must remove those that you don't want to become a 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.

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**NOTE** To include all layouts when you add sheets to a drawing set, make sure the Include Layouts When Adding Sheets option is selected on the shortcut menu or in the Publish dialog box.

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- **Include model layouts.** If an uninitialized (the paper size is not defined in the page setup or is set to 0 x 0) model layout is included, it will be marked as Uninitialized in the Status column on the sheet list. It can be plotted if a page setup override is applied.

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**NOTE** To include the Model when you add sheets to a drawing set, make sure the Include Model When Adding Sheets option on the shortcut menu is selected.

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- **Remove sheets.** To remove sheets from the list, select one or more sheets in the list. Click the Remove Sheets button. To remove all sheets, right-click. Click Remove All.
- **Reorder sheets.** To reorder the sheets up or down one position in the list, select a sheet. Click either the Move Sheet Up or Move Sheet Down button. Sheets in the drawing set are viewed or plotted in the order shown in the list.
- **Rename sheets.** To rename a sheet, select it in the list and right-click, Click Rename Sheet. Enter the new sheet name.
- **Change page setups.** To change the page setup for a layout, select the sheet. In the Page Setup list, select a named page setup from the list, or select one or more sheets in the list and right-click. Click Change Page Setup. In the Page Setup list, select a page setup or select Import to import page setups from another drawing or template. In the Import Page Setups dialog box, select a drawing with one or more page setups. Click Import. In the Page Setup list, select a named page setup.

**NOTE** It is important that you change the page setup for each layout depending on your desired output. Only model space page setups can be applied to model space sheets, and only paper space page setups can be applied to paper space sheets.

- **Copy sheets.** To copy one or more drawing sheets, select the sheets in the list and right-click. Click Copy Selected Sheets. The copied drawing sheets are added and highlighted at the end of the sheet list. When a sheet is copied, its name is created by the addition of *-Copy(n)* at the end of the original sheet name. For example, if you create one copy of a sheet called *Plumbing*, the copied sheet is called *Plumbing-Copy(1)*. Each time you copy the same sheet, the *n* is incremented by 1. By creating copies of a sheet, you can have different page setups and other settings for the same sheet.

3. When your list of drawing sheets is assembled and configured the way you want for your paper or plot file drawing set, click the Save Sheet List button.
4. In the Save List As dialog box, in the File Name box, enter a name for the drawing set list. Click Save.

   The drawing set list is saved as a DSD (Drawing Set Descriptions) file.
5. In the Publish dialog box, under Publish To, select Plotter Named in Page Setup.
6. Click Publish to start the process.

   If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress: paper drawings are being plotted or plot files are being created.
7. To view information about the processed publish job, right-click the plotter icon on the right side of the status bar. Click View Plot and Publish Details.

Standard toolbar

# Publish an Electronic Drawing Set

You can publish an electronic drawing set in the Design Web Format.

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.

An electronic drawing set is saved as a single, multisheet DWF file and can be shared with customers, suppliers, or people within your own company who may need the drawings for examination or for their records.

You can also send the published electronic drawing set as an email attachment, share it using a project collaboration site such as Autodesk® Buzzsaw®, or post it to a website. Using Autodesk DWF Viewer, you can view or plot only the layouts you need.

You can also publish the drawing set as individual, single-sheet DWF files for each sheet.

When you publish to a DWF file, you use the *DWF6 ePlot.pc3* plotter configuration file. You can use the default *DWF6 ePlot.pc3* 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 *DWF6 ePlot.pc3* file, all future plotting and publishing of DWF files will be affected.

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**WARNING** Before you make changes to the original *DWF6 ePlot.pc3* file, make sure you have backed up a copy in case you want to return to the default settings.

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**See also:**

"Set Publish Options" on page 940

### To create an electronic drawing set

1. With a drawing open, click File menu ➤ 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. 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, make sure the Include Layouts When Adding Sheets option is selected on 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, make sure the Include Model When Adding Sheets option is selected either on the shortcut menu or in the Publish dialog box.

- **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** It is important that you change the page setup for each layout depending on your desired output. Only model space page setups can be applied to model space sheets, and only paper space page setups can 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.
5. In the Publish dialog box, under Publish To, select DWF File.
6. In the Select DWF File dialog box, enter a file name. Click Select to provide the name and destination for the DWF file.

   **NOTE** You can also enter a URL so that the DWF file is uploaded to an FTP or HTTP site.
7. Click Publish 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.
8. 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.
9. If you have background publishing enabled you can view the published DWF file. Right-click the plotter icon on the right side of the status. Click View DWF File.

Standard toolbar

# Publish a Sheet Set

From the Sheet Set Manager, you can easily publish an entire sheet set, a subset of a sheet set, or a single sheet. If you want to publish a sheet set that you've set up in the Sheet Set Manager, it's much quicker to publish the sheet set directly from the Sheet Set Manager, rather than starting from the Publish dialog box.

When you publish from the Sheet Set Manager, you can publish an electronic sheet set by publishing to a DWF file, or you can publish a paper set by publishing to the plotter named in the page setup that is associated with each drawing sheet. You can also publish your sheets using a page setup that is saved in the page setup overrides DWT file associated with the sheet set. This page setup overrides the current page setup settings for the individual publish job.

When you open the Publish dialog box from the Sheet Set Manager, the Publish dialog box automatically lists the sheets you selected in the sheet set. You can then modify the sheet set for publishing.

**See also:**

"Use Named Page Setups with Sheet Sets" on page 867
"Work with Sheets in a Sheet Set" on page 233
"Set Publish Options" on page 940

### To publish a sheet set to a DWF file

1. In the Sheet Set Manager, under Sheets, select the Sheet Set node or name.
2. In the upper-right corner of the Sheet Set Manager, click the Publish to DWF button.

**NOTE** If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress. Right-click this icon for options to view the DWF file or to view information about the publish job.

Standard toolbar

Right-click a sheet set node or name. Click Publish ➤ Publish to DWF.

### To publish a subset of a sheet set or an individual sheet to a DWF file

1. In the Sheet Set Manager, under Sheets, select the node or name of a subset or an individual sheet.
2. In the upper-right corner of the Sheet Set Manager, click the Publish to DWF button.

**NOTE** If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress. Right-click this icon for options to view the DWF file or to view information about the publish job.

Standard toolbar

Right-click the node or name of a subset or an individual sheet. Click Publish ➤ Publish to DWF.

**To publish sheets in a sheet set to the plotter named in the sheet's page setup**

1. In the Sheet Set Manager, under Sheets, select the node or name of a sheet set, a subset, or an individual sheet.
2. In the upper-right corner of the Sheet Set Manager, click the Publish button. Click Publish to Plotter.

**NOTE** If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress. Right-click this icon for options to view the DWF file or to view information about the publish job.

Standard toolbar

Right-click the node or name of a sheet set, subset, or an individual sheet. Click Publish ➤ Publish to Plotter.

**To publish sheets in a sheet set using a page setup from the page setup overrides file**

1. In the Sheet Set Manager, under Sheets, select the node or name of a sheet set, subset, or an individual sheet.

2 In the upper-right corner of the Sheet Set Manager, click the Publish button. Click Publish Using Page Setup Override ➤ <*filename.dwt*>. (All the named page setups that are saved in the page setup override DWT file are listed.)

The settings in the selected page setup override file take precedence over (override) the page setup settings currently set for the selected sheets. The page setup override settings apply only for this individual publish job.

**NOTE** If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress. Right-click this icon for options to view the DWF file or to view information about the publish job.

Standard toolbar

Right-click the node or name of a sheet set, subset, or an individual sheet. Click Publish ➤ Publish Using Page Setup Override ➤ <*filename.dwt*>.

**To include a plot stamp on published sheets from a sheet set**

■ In the upper-right corner of the Sheet Set Manager, click the Publish button. Click Include Plot Stamp.

A plot stamp will be included on all published sheets.

**NOTE** To change the plot stamp settings, click the Publish button. Click Plot Stamp Settings.

Standard toolbar

Right-click the node or name of a sheet set, subset, or an individual sheet. Click Publish ➤ Include Plot Stamp.

**To set options for publishing sheet sets**

1. In the upper-right corner of the Sheet Set Manager, click the Publish button. Click Sheet Set Publish Options.

2. In the Sheet Set Publish Options dialog box, modify the settings as required.
3. Click OK.

Standard toolbar

Right-click the node or name of a sheet set, subset, or an individual sheet. Click Publish ➤ Sheet Set Publish Options.

**To open the Publish dialog box from the Sheet Set Manager and modify the sheet set for publishing**

1. In the Sheet Set Manager, under Sheets, select the node or name of a sheet set, subset, or an individual sheet.
2. In the upper-right corner of the Sheet Set Manager, click the Publish button. Click Publish Dialog Box.

   The Publish dialog box is displayed and lists the sheets you selected in the Sheet Set Manager.
3. In the Publish dialog box, modify the sheet list and settings as required.
4. Click Publish to publish the modified sheet set.

**NOTE** If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress. Right-click this icon for options to view the DWF file or to view information about the publish job.

Standard toolbar

Right-click the node or name of a sheet set, subset, or an individual sheet. Click Publish ➤ Publish Dialog Box.

# Republish a Drawing Set

After drawings have been updated, you may want to republish a drawing set for viewing or plotting. You can easily republish a sheet set (DST file) or a collection of drawing sheets that you've saved a description of as a DSD (Drawing Set Descriptions) file. You can also load and republish BP3 (Batch Plot) files.

**To republish a drawing set**

1. Click File menu ➤ Publish.

   If you have a drawing open, all the layouts in your drawing are shown in the sheet list in the Publish dialog box. Right-click in the sheet list. Click Remove All to delete the layouts from the list of drawing sheets.

2. In the Publish dialog box, click the Load Sheet List button.

3. In the Load List of Sheets dialog box, select the location of the DSD file or the BP3 file. Click Load.

   The drawing sheets in the saved drawing set are displayed in the sheet list in the Publish dialog box.

4. In the Publish dialog box, do one of the following:

   - If you want to publish the drawing set to a DWF file, under Publish To, select DWF file.
   - If you want to publish the drawing set to a plotter or printer, under Publish To, select Plotter Named in Page Setup.

5. Click Publish.

   If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress.

6. If you have background publishing enabled, you can view information about the processed publish job. Right-click the plotter icon on the right side of the status bar. Click View Plot and Publish Details. The information in the Plot and Publish Details dialog box is also saved to the plot and publish log file.

Standard toolbar

# View a Published Electronic Drawing Set

A published electronic drawing set can be viewed and plotted with Autodesk™ DWF™ Viewer.

A published drawing set is the digital equivalent of paper plots created from your original drawings. The drawing set, which is saved as a Design Web Format™ (DWF™ ) file, can be viewed or plotted by anyone using Autodesk DWF Viewer. With Autodesk DWF Viewer, you can open, view, and print all DWF file formats and other raster format images. You can pan, zoom, and view individual drawing sheets and viewports. You can also view layer information, sheet and sheet set properties, block information and attributes, and custom properties, if they are included in the DWF file. As you move your cursor across DWF geometry in the viewer, objects with associated data are displayed with a red highlighting effect.

Recipients of drawing sets in DWF format don't have to own or know the program. From anywhere in the world, they can view and print high-quality layouts using Autodesk DWF Viewer.

Autodesk DWF Viewer runs as a stand-alone application or embedded in any application that supports ActiveX controls, such as Microsoft® Internet Explorer.

For more information about using Autodesk DWF Viewer, see Help in Autodesk DWF Viewer.

**To view the most recently published DWF file in Autodesk DWF Viewer**

- 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 only if you have background publishing enabled.

# Set Publish Options

You can set options for publishing, such as output file location, DWF type, multisheet DWF name options, DWF security (password protection), and whether or not to include layer information. You can also decide what types

of information to reveal in your published DWF files. You can include the following types of metadata:

- Sheet set properties (must publish using the Sheet Set Manager)
- Sheet properties (must publish using the Sheet Set Manager)
- Block standard properties and block custom properties and attributes
- Properties contained in custom objects

You use a block template (BLK) file to determine which blocks and properties to include in your published DWF 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 save them to just the DSD file.

**To change the default output location for published DWF and plot files**

1. Click File menu ➤ Publish.
2. In the Publish dialog box, under Publish To, click either Plotter Named in Page Setup or DWF File, and then click Publish Options.
3. In the Publish Options dialog box, under Default Output Location (DWF and Plot-to-File), do one of the following:
   - Click Location and make a selection from the drop-down list.
   - Click the [...] button, and in the Select a Folder for Generated Files dialog box, select a folder. Click Select.
4. In the Publish Options dialog box, click OK.

Standard toolbar

**To specify the DWF file output type to single-sheet DWF files or a single multi-sheet DWF file**

1. Click File menu ➤ Publish
2. In the Publish dialog box, under Publish To, select the DWF File, and then click Publish Options.

3 In the Publish Options dialog box, under General DWF Options, DWF Type, make a selection from the drop-down list as follows:

- **Single-sheet DWF.** Specifies that an individual single-sheet DWF file is created for each sheet.
- **Multi-sheet DWF.** Specifies that one multi-sheet DWF file is created.

4 Click OK.

5 In the Publish dialog box, either continue with publishing tasks or close the dialog box.

Standard toolbar

**To specify a name for multi-sheet DWF files**

1 Click File menu ➤ Publish

2 In the Publish dialog box, under Publish To, click DWF File, and then click Publish Options.

3 In the Publish Options dialog box, under General DWF Options, DWF Type, select Multi-sheet DWF from the drop-down list.

4 In the Publish Options dialog box, under Multi-sheet DWF Options, DWF Naming, select Specify Name from the drop-down list.

5 Click OK.

6 In the Publish dialog box, either continue with publishing tasks or close the dialog box.

Standard toolbar

**To specify to be prompted for a name for a multi-sheet DWF file**

1 Click File menu ➤ Publish

2 In the Publish dialog box, under Publish To, click DWF File, and then click Publish Options.

3 In the Publish Options dialog box, under General DWF Options, DWF Type, select Multi-sheet DWF from the drop-down list.

4 In the Publish Options dialog box, under Multi-sheet DWF Options, select Prompt for Name to be prompted for a file name every time you publish a DWF file.

5 Click OK.

6 In the Publish dialog box, either continue with publishing tasks or close the dialog box.

Standard toolbar

**To specify that DWF files are published with password security**

1 Click File menu ➤ Publish.

2 In the Publish dialog box, under Publish To, click DWF File, and then click Publish Options.

3 In the Publish Options dialog box, under General DWF Options, Password Protection, select one of the following from the drop-down list:

- Select Specify Password, and under Password, enter a password to use for the DWF file.
- Select Prompt for Password. You will be prompted for a password or phrase to open the published DWF file.

DWF passwords are case-sensitive. The password or phrase can consist of letters, numbers, punctuation marks, or non-ASCII characters.

**WARNING** If you lose or forget the password, it cannot be recovered. Keep a list of passwords and their corresponding DWF file names in a safe place.

4 Click OK.

5 In the Publish dialog box, either continue with publishing tasks or close the dialog box.

Standard toolbar

**To include layer information in a published DWF file**

1 Click File menu ➤ Publish.

2 In the Publish dialog box, under Publish To, click DWF File, and then click Publish Options.

3 In the Publish Options dialog box, under DWF Data Options, Layer Information, click to display the drop-down list, and select Include.

**NOTE** By default, Layer Information is set to Don't Include, to reduce publishing time. If you change the setting to include layer information, you can turn individual layers on and off when you view or print the DWF file.

4 Click OK.

5 In the Publish dialog box, either continue with publishing tasks or close the dialog box.

Standard toolbar

**To include block information in a published DWF file**

1 Click File menu ➤ Publish.

2 In the Publish dialog box, under Publish To, click DWF File, and then click Publish Options.

3 In the Publish Options dialog box, under DWF Data Options, Block Information, click to display the drop-down list, and select Include.

**NOTE** By default, Block Information is set to Don't Include. If you change the setting to include block information, you can use the viewer to view or print block property and attribute information in the DWF file.

4 Click OK.

5 In the Publish dialog box, either continue with publishing tasks or close the dialog box.

Standard toolbar

**To include block template information in a published DWF 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 File menu ➤ Publish.
2. In the Publish dialog box, under Publish To, click DWF File. Click Publish Options.
3. In the Publish Options dialog box, under DWF Data Options, Block Information, click to display the drop-down list. Select Include.
4. In the Publish Options dialog box, under DWF Data Options, Block Template File, click to display the drop-down list. Select the block template (BLK) file that contains the properties and attribute definitions you want to make available in your published DWF file. Click OK.

   **NOTE** The list also contains the Create and Edit options. Create opens the Publish Block Template dialog box where you can create a new block template file. Edit opens the Edit Publish Template dialog box where you can select an existing block template file to modify.

5. Click OK.
6. In the Publish dialog box, either continue with publishing tasks or close the dialog box.

Standard toolbar

**To create a block template (BLK) file**

1. Click File menu ➤ Publish.
2. In the Publish dialog box, under Publish To, click DWF File, and then click Publish Options.
3. In the Publish Options dialog box, under DWF Data Options, Block Information, click to display the drop-down list, and select Include.
4. Under DWF Data Options, Block Template File, click to display the drop-down list, and select Create.

   The Publish Block Template dialog box opens with the Block Source Drawing area displaying the current drawing.

**NOTE** The working set of drawings for this list is completely independent of the drawings to include in the Publish operation.

5 Do one of the following:

- If this drawing contains the block properties you want to include in the published DWF 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 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 file. Click Scan for Blocks.

  The DWG file is scanned for all unique block definitions and their associated properties and attributes.

  **NOTE** By default, block settings for nested blocks and blocks in xrefs are included. If you don't want to include these block settings, click Options in the Publish Block Template dialog box and clear the related options.

6 In the Publish Block Template dialog box, under Block Data to Publish, Unique Blocks From Source Drawings, Check Blocks to Publish, select the block names you want to include in the published DWF files. You can right-click to select or clear all boxes.

7 Under Block Data to Publish, Properties of Selected Blocks, Check Properties to Publish, select the properties you would like to include in your published DWF 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 file; no property information is included.

8 Click Save to name and save the block template file. Click OK.

The new block template file is now available under DWF Data Options in the Publish Block Template dialog box.

9 In the Publish dialog box, either continue with publishing tasks or close the dialog box.

Standard toolbar

**To edit a block template (BLK) file**

1. Click File menu ➤ Publish.
2. In the Publish dialog box, under Publish To, click DWF File, and then click Publish Options.
3. In the Publish Options dialog box, under DWF Data Options, Block Information, click to display the drop-down list, and select Include.
4. Under DWF Data Options, Block Template File, click to display the drop-down list, and select Edit.
5. In the Select Block Template dialog box, click the Block Template (BLK) file you want to edit, and then click Select.

   **NOTE** The working set of drawings for this list is completely independent of the drawings to include in the Publish operation.

6. In the Publish Block Template dialog box, click Scan for Blocks.

   The program scans the DWG files for all unique block definitions and their associated properties and attributes.

   **NOTE** By default, block settings for nested blocks and blocks in xrefs are included. If you don't want to include these block settings, click Options in the Publish Block Template dialog box and clear the related options.

7. Under Block Data to Publish, Unique Blocks From Source Drawings, Check Blocks to Publish, select or clear the block names you want to include in the published DWF files.
8. Under Block Data to Publish, Properties of Selected Blocks, Check Properties to Publish, select or clear the properties you want to include in your published DWF 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 file; no property information is included.

9. Click Save. Click OK.

   The modified block template file is now available under DWF Data Options in the Publish Block Template dialog box.

10 In the Publish dialog box, either continue with publishing tasks or close the dialog box.

Standard toolbar

# 3D DWF Publishing

Using 3D DWF Publishing, you can create and publish Design Web Format™ (DWF™ ) files of your three-dimensional models. As a technology preview in AutoCAD® 2006, the 3DDWFPUBLISH command is a default feature in all network installations and an optional feature in stand-alone installations.

With 3D DWF Publishing, you can generate DWF files of your three-dimensional models with nearly the same visual fidelity as your original DWG files. 3D DWF Publishing creates a single-sheet DWF file which contains just model space objects. Access to the 3D DWF Publishing feature is limited to command line interaction. As a technology preview, there are some known limitations. Please refer to the Readme file.

Recipients of 3D DWF files can view and print them using Autodesk® DWF™ Viewer. For more information about using Autodesk DWF Viewer, see the Autodesk DWF Viewer Help system.

### Increase the Smoothness of 3D DWF Models

The smoothness of 3D DWF models can be increased by changing the value of the FACETRES system variable. For 3DDWFPUBLISH, two special settings of FACETRES, 9 and 10, increase smoothing. These values multiply the effect of FACETRES.

**NOTE** Higher values of FACETRES may increase 3D DWF file sizes.

### To publish a 3D DWF file

**NOTE** The 3DDWFPUBLISH command is available only if you have installed the 3D DWF Publishing feature.

1 On the command line, enter **3ddwfpublish** .

2 In the 3D DWF Publish dialog box, under DWF File Name, change the name and location of the DWF file to be saved, or use the [...] button to navigate to a new location for the 3D DWF file.

3 Under Objects to Publish, click one of the following:

- **All Model Space Objects.** All model space objects are published to the 3D DWF file.
- **Selected Model Space Objects.** You are prompted to create a selection set of model space objects which are published to the 3D DWF file.

4 If your drawing contains xrefs, the Group By Xref Hierarchy option is selected by default. Clear the check box if you do not want to display the objects grouped by xref hierarchy in the published DWF file.

5 Under Group Individual Objects By, select one of the following options to group individual objects in the DWF file for viewing. Click OK.

- **Layer.** Groups individual objects by layer.
- **Object Type.** Groups individual objects by object type (for example, by block).

6 (Optional) Click Yes to open the Autodesk DWF Viewer and view the published 3D DWF file.

### To install the 3D DWF Publishing feature

- In Windows Control Panel, double-click Add or Remove Programs.
- In the Add or Remove Programs dialog box, select AutoCAD 2006. Click Change.
- In the AutoCAD 2006 Setup dialog box, click Add or Remove Features. Click Next.
- On the Please Select Which Features You Would Like to Install page, click the red X next to 3D DWF Publish. Select Entire Feature Will Be Installed on Local Hard Drive.
- On the Install Optional Tools page, the Install 3D DWF Publish option is selected. Click Next.
- On the Start Installation page, click Next to start the installation.
- On the AutoCAD 2006 Has Been Successfully Installed page, click Finish.

**To view and print a published 3D DWF file**

- If you want to view a 3D DWF file, do one of the following:
  - Immediately after the 3D DWF file has been published, click Yes to the question "Would you like to view it now?" The Autodesk DWF Viewer opens the 3D DWF file.
  - In Windows Explorer, double-click the 3D DWF file to open the Autodesk DWF Viewer and view the file.
  - Open the Autodesk DWF Viewer. Click File menu ➤ Open. Select the 3D DWF file you want to view.
- If you want to print a 3D DWF file, in Autodesk DWF Viewer, click File menu ➤ Print. For more information, see the Autodesk DWF Viewer Help system.

# Configure a DWF6 Driver (Advanced)

You are not likely to change the settings in the DWF6 ePlot Properties dialog box often. However, when you need to make some subtle changes to your DWF6 ePlot configuration, you can do so through the DWF6 ePlot Properties dialog box. You can specify settings for color depth, display resolution, file compression, handling of fonts, pen settings, and other options. When you create DWF6 files, you use a plotter configuration file that uses a DWF6 driver model. You must use the *DWF6 ePlot.pc3* file to plot DWF6 files.

## Overview of Creating or Modifying a DWF6 Configuration File

Publish uses only the DWF6 *ePlot.pc3* plotter configuration file. You can use the default DWF6 *ePlot.pc3* plotter configuration file as installed, or you can make changes to it using the Properties button in the Plot dialog box. This invokes the pc3 editor, which allows you to make changes directly to the DWF6 *ePlot.pc3* file. Although you can save DWF6 *ePlot.pc3* files under other names for use with the Plot command, no other name will work with the Publish command. Any changes you make to the DWF6 *ePlot.pc3* file are used whenever you plot or publish DWF6 files until you change the settings of your DWF6 *ePlot.pc3* file again.

**NOTE** If you are going to make changes to the DWF6 *ePlot.pc3* file, make a backup copy of the file first in case you want to use the default settings later. If necessary, the default DWF6 *ePlot.pc3* file can be re-created with the Add-A-Plotter wizard.

These settings may change file size and plot quality, depending on the content of the source drawing for the Design Web Format™ (DWF™ ) file. You can specify the following settings when you edit the DWF6 *ePlot.pc3* configuration file:

- Color depth
- Display resolution
- Compression options
- Inclusion and handling of fonts
- Background color
- Virtual pen settings and patterns
- Inclusion of layer information
- Inclusion of a paper boundary
- Inclusion of a saved preview

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**NOTE** DWF files intended for plotting should be configured with a white background color. If the background color is set to black, color 7 objects will be plotted as white. For all other background colors, color 7 objects will be plotted as black.

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**To create a plotter configuration file for DWF file output**

1. Click File menu ➤ Plotter Manager.
2. Double-click the Add-A-Plotter wizard.
3. On the Add Plotter-Introduction page, click Next.
4. On the Add Plotter-Begin page, select My Computer. Click Next.
5. On the Plotter Model page, under Manufacturers, select Autodesk ePlot (DWF). Under Models, select the DWF6 ePlot file that you want to create. Click Next.
6. (Optional) If you want to import a pre-existing plotter configuration, on the Import PCP or PC2 page, click Import File. Select a PCP or PC2 file to import. Click Import.
7. Click Next.
8. On the Ports page, select Plot to File. Click Next.

9 On the Plotter Name page, enter a name for the plotter configuration file. Click Next.

10 On the Finish page, click Finish.

A new plotter configuration file (PC3) is created.

**To specify or modify settings for plotted DWF files**

1 Click File menu ➤ Plot.

2 In the Plot dialog box, under Printer/Plotter, in the Name list, select a DWF plotting device. Click Properties.

3 In the Plotter Configuration Editor, Device and Document Settings tab, select Custom Properties in the tree window.

4 Click Custom Properties.

5 In the DWF6 ePlot Properties dialog box, select the options you want. Click OK.

6 In the Plotter Configuration Editor, click OK.

7 In the Changes to a Printer Configuration dialog box, do one of the following. Then click OK.

- Select Apply Changes for the Current Plot Only to specify a one-time override of the configuration settings that isn't saved to the ePlot configuration file.
- Select Save Changes to the Following File to save the configuration changes to the DWF configuration file.

8 In the Browse for Plot File dialog box, in the Save In location list, select a plotting location for the DWF file, and click Save. Click OK.

Standard toolbar

**To set color depth for plotted DWF files**

1 Click File menu ➤ Plot.

2 In the Printer/Plotter area, in the Name list, select a DWF plotting device. Click Properties.

3 In the Plotter Configuration Editor, Device and Document Settings tab, expand the Graphics node in the tree window.

4 Click Vector Graphics.

5 In the Color Depth area, select a color depth. Click OK.

6 In the Changes to a Printer Configuration dialog box, do one of the following. Then click OK.

- Select Apply Changes for the Current Plot Only to specify a one-time override of the configuration settings that isn't saved to the DWF configuration file.
- Select Save Changes to the Following File to save the configuration changes to the DWF configuration file.

7 In the Browse for Plot File dialog box, in the Save In location list, select a plotting location for the DWF file, and click Save. Click OK.

Standard toolbar

# Set the DWF File Resolution

You can specify the pixel resolution for vector and raster graphics of the DWF6 files that you create. The higher the resolution, the greater the precision but the larger the file size.

**WARNING** Raster and gradient resolution settings can never exceed the vector setting.

The following list shows the default values for vector and raster resolutions:

- Vector resolution: 1200 dpi
- Custom vector resolution: 40000 dpi
- Gradient resolution: 200 dpi
- Custom gradient resolution: 200 dpi
- Color and grayscale resolution: 200 dpi
- Custom color and grayscale resolution: 200 dpi
- Black and white resolution: 400 dpi
- Custom black and white resolution: 400 dpi

When you create DWF files intended for plotting, select a resolution to match the output of your plotter or printer. High resolutions (over 2400 dpi) are for viewing. For example, when you create DWF files of drawings that contain a lot of detail, such as a topographical map of a large region, use a higher resolution setting for greater detail in the DWF file. Use extreme resolutions (over 40,000 dpi) only when necessary; they may produce very large files. As you increase the resolution setting, raster image quality increases, the speed of printing decreases, and memory requirements increase.

As an example of the difference between DWF resolution settings, consider a map of the world that you want to output as a DWF file for viewing. With a medium resolution setting, you might be able to zoom to a level of detail approximately the size of the state of California on the world map. With a high resolution setting, you might be able to zoom to a level of detail the size of a city. With an extreme resolution setting, you might be able to zoom to a level of detail the size of a building.

**To specify the DWF file resolution**

1. Click File menu ➤ Plot.
2. In the Printer/Plotter area, in the Name list, select a DWF plotting device. Click Properties.
3. In the Plotter Configuration Editor, Device and Document Settings tab, select Custom Properties from the tree window.
4. In the Access Custom Dialog area, click Custom Properties.

   **NOTE** You can enter any whole number between 150 and 100,000,000 for color and grayscale resolution settings as long as the number does not exceed the current vector resolution setting.

5. In the DWF6 ePlot Properties dialog box, in the Vector and Gradient Resolution (Dots Per Inch) area, select Vector and Gradient Resolution settings from the list, or select Custom and enter custom settings. Click OK.
6. In the DWF6 ePlot Properties dialog box, in the Raster Image Resolution (Dots Per Inch) area, select Color and Grayscale Resolution settings or Black and White Resolution settings from the list, or enter Custom settings. Click OK.
7. In the Plotter Configuration Editor, click OK.

8 In the Changes to a Printer Configuration dialog box, do one of the following. Then click OK.

- Select Apply Changes for the Current Plot Only to specify a one-time override of the configuration settings that isn't saved to the DWF configuration file.
- Select Save Changes to the Following File to save the configuration changes to the DWF configuration file.

9 In the Browse for Plot File dialog box, in the Save In location list, select a plotting location for the DWF file, and click Save. Click OK.

Standard toolbar

# Set the DWF File Compression

By default, the DWF6 files that you create are in a compressed binary format. The compression does not result in any loss of data; this is the recommended output for most DWF files. You can also create zipped ASCII encoded 2D stream (plain text) files. You specify these settings when you create or edit a DWF configuration file.

**To specify the DWF file compression**

1 Click File menu ➤ Plot.

2 In the Printer/Plotter area, in the Name list, select a DWF plotting device. Click Properties.

3 In the Plotter Configuration Editor, Device and Document Settings tab, select Custom Properties from the tree window.

4 In the Access Custom Dialog area, click Custom Properties.

5 In the DWF6 ePlot Properties dialog box, in the Additional Output Settings and in the DWF Format area, specify a file compression option. Click OK.

6 In the Plotter Configuration Editor, click OK.

7 In the Changes to a Printer Configuration dialog box, do one of the following. Then click OK.

- Select Apply Changes for the Current Plot Only to specify a one-time override of the configuration settings that isn't saved to the DWF6 configuration file.
- Select Save Changes to the Following File to save the configuration changes to the DWF configuration file.

8 In the Browse for Plot File dialog box, in the Save In location list, select a plotting location for the DWF file, and click Save. Click OK.

Standard toolbar

# Set Font Handling for the DWF File

When you create DWF files, you can specify how fonts are handled and included in the DWF6 file. By default, font handling is set to Capture Some in the DWF6 ePlot Properties dialog box; you can specify which fonts are eligible for capture in your DWF file. This is the recommended option.

**NOTE** DWF file size can be affected by the font-handling settings as well as by the amount of text and the number and type of fonts used in the DWF file. If the size of your DWF file seems too large, try changing the font-handling settings.

**To specify font handling for the DWF file**

1 Click File menu ➤ Plot.

2 In the Printer/Plotter area, in the Name list, select a DWF plotting device. Click Properties.

3 In the Plotter Configuration Editor, Device and Document Settings tab, select Custom Properties from the tree window.

4 In the Access Custom Dialog area, click Custom Properties.

5 In the DWF6 ePlot Properties dialog box, in the Font Handling area, select a font capture option. Click OK.

6 In the Plotter Configuration Editor, click OK.

7 In the Changes to a Printer Configuration dialog box, do one of the following. Then click OK.

- Select Apply Changes for the Current Plot Only to specify a one-time override of the configuration settings that isn't saved to the DWF6 configuration file.
- Select Save Changes to the Following File to save the configuration changes to the DWF configuration file.

8 In the Browse for Plot File dialog box, in the Save In location list, select a plotting location for the DWF file, and click Save. Click OK.

Standard toolbar

**To edit the list of fonts eligible for capture in the DWF file**

1 Click File menu ➤ Plot.

2 In the Printer/Plotter area, in the Name list, select a DWF plotting device. Click Properties.

3 In the Plotter Configuration Editor, Device and Document Settings tab, select Custom Properties from the tree window.

4 In the Access Custom Dialog area, click Custom Properties.

5 In the DWF6 ePlot Properties dialog box, in the Font Handling area, select Capture Some.

6 Select Edit Font List.

---

**NOTE** Capturing fonts in the DWF file causes the file size to increase. To minimize file size, the TrueType fonts that are common to all Microsoft® Windows® platforms are by default not selected in the list. Even though these fonts have no check marks beside them, they are shown in the DWF6 file if you are using a Windows operating system. You should select all other TrueType fonts that you have installed to ensure that they are captured for embedding in the DWF6 file. Only the fonts that are needed are embedded in the file.

---

7 In the Available TrueType Fonts dialog box, select the fonts that you want to be eligible for capture in the DWF file. Click OK.

8 In the DWF6 ePlot Properties dialog box, click OK.

9 In the Plotter Configuration Editor dialog box, click OK.

10 In the Browse for Plot File dialog box, in the Save In location list, select a plotting location for the DWF file, and click Save. Click OK.

Standard toolbar

# Edit Pen Patterns for the DWF File

In the Edit Pen Set dialog box, you can specify pen pattern, width, shape, effect, and other settings such as scaling and global pen widths. The changes you make in the Edit Pen Set dialog box are saved in the plotter configuration file.

**NOTE** To edit pen patterns when using DWF6 ePlot, you must select 255 Virtual Pens as your color depth.

**To edit the Edit Pen Set dialog box for the DWF file**

1 Click File menu ➤ Plot.

2 In the Printer/Plotter area, in the Name list, select a DWF plotting device. Click Properties.

3 In the Plotter Configuration Editor, Device and Document Settings tab, expand the Graphics node in the tree window. Select Vector Graphics. In the Color Depth Area, select 255 Virtual Pens as your color depth, and then select Custom Properties from the tree window.

4 In the Access Custom Dialog area, click Custom Properties.

5 In the DWF6 ePlot Properties dialog box, click Edit Patterns.

6 In the Edit Pen Set dialog box, right-click a field.

7 Select from the menu of common settings, or click Properties to display the Pen Properties dialog box where you can modify the attributes for each individual pen set.

8 In the Pen Properties dialog box, click OK.

9 In the Edit Pen dialog box, click OK.

10 In the DWF6 ePlot Properties dialog box, click OK.

11 In the Plotter Configuration Editor dialog box, click OK.

12 In the Changes to a Printer Configuration File dialog box, click one of the following:

- Apply Changes for the Current Plot Only.
- Save Changes to the Following File. (Enter the file name.)

13 In the Browse for Plot File dialog box, in the Save In location list, select a plotting location for the DWF file, and click Save. Click OK.

Standard toolbar

# Part 8
# Share Data Between Drawings and Applications

Chapter 29 Reference Other Drawing Files (Xrefs)
Chapter 30 Link and Embed Data (OLE)

# Reference Other Drawing Files (Xrefs)

# 29

You can attach an entire drawing to the current drawing as an external reference (xref). With xrefs, changes 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.

## In this chapter


- Overview of External References
- Attach, Update, and Bind External References
- Detach External References
- Edit External References and Blocks in Place
- Set Paths to Externally Referenced Drawings
- Resolve External Reference Errors
- Increase Performance with Large Xrefs

# Overview of External References

By using xrefs, 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 users. 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 xref 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 xrefs permanently with your 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 61

# Attach, Update, and Bind External References

You can perform several operations on referenced drawing files including attaching, updating, and detaching them.

## Attach External References

When you attach a drawing 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.

The program treats an xref as a type of block definition with some important differences. When you insert a drawing as a block reference, it is stored in the drawing and is not updated if the original drawing changes. When you attach a drawing 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.

A drawing can be attached as an xref to multiple drawings at the same time. Conversely, multiple drawings can be attached as external references to a single drawing.

The saved path used to locate the xref can be an absolute (fully specified) path, a relative (partially specified) path, or no path.

If an xref contains any variable block attributes, The program ignores them.

**NOTE** Xrefs must be model space objects. They can be attached at any scale, location, and rotation.

### Notification of Attached Xrefs

An xref icon is displayed in the lower-right corner of the application window (the status bar tray) when xrefs are attached to the drawing.

When one or more xrefs are not found or if any of the xrefs need reloading, an exclamation point is added to the Xref icon. If you click the Xref icon, the Xref Manager is displayed.

### Tools for Attaching Xrefs

To attach an xref, use  to display the Xref Manager or use .

You can also use DesignCenter™ to attach xrefs to a drawing. Use DesignCenter for simple attachments, previewing external 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.

### Visibility, Color, and Linetype

You can control the visibility, color, linetype, and other properties of an xref's layers and make these changes temporary or permanent. If the  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.

### Xref Clipping Boundaries

Drawings can include xrefs that are clipped. If you want to see the clipping boundary, you can turn on the  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:**


"DesignCenter" on page 61
"Nest and Overlay External References" on page 967
"Clip External References and Blocks" on page 970
"Set Paths to Externally Referenced Drawings" on page 983


### To attach an xref

1. Click Insert menu ➤ External Reference.
2. In the Select Reference File dialog box, select the file you want to attach. Click Open.
3. In the 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.

Reference toolbar

### To attach or overlay an xref using DesignCenter

1. Click Tools menu ➤ DesignCenter.
2. In the content area or in the Search dialog box, locate the xref you want to attach or overlay.
3. Right-click. Drag the xref 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.

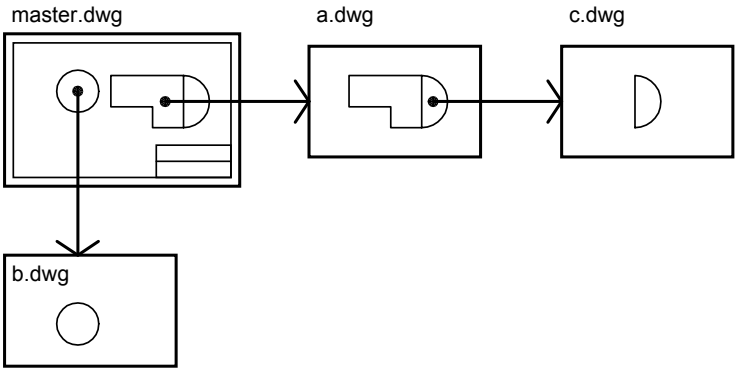
8 Click OK.

Standard toolbar

## Nest and Overlay External References

Xrefs can be nested within other xrefs: that is, you can attach an xref that contains another xref. You can attach as many copies of an xref as you want, and each copy can have a different position, scale, and rotation.

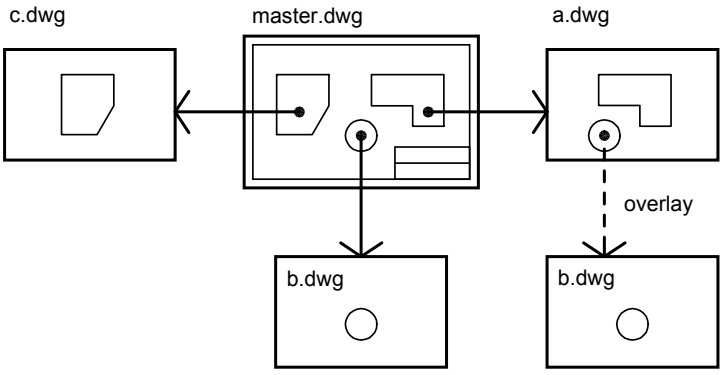
In the following illustration, *master.dwg* references *a.dwg* and *b.dwg*. Drawing *a.dwg* references *c.dwg*. In *master.dwg*, *c.dwg* is a nested xref.




You can also overlay an xref on your drawing. Unlike an attached xref, an overlaid xref is *not* included when the drawing is itself attached or overlaid as an xref to another drawing. Overlaid xrefs are designed for data sharing in a network environment. By overlaying an xref, you can see how your drawing

relates to the drawings of other groups without changing your drawing by attaching an xref.

In the following illustration, several people are working on drawings referenced by *master.dwg*. The person working on *a.dwg* needs to see the work being completed by the person working on *b.dwg*, but does not want to xref *b.dwg* because it would then appear twice in *master.dwg*. Instead, the person overlays *b.dwg*, which is not included when *a.dwg* is referenced by *master.dwg*.




### Relative Saved Paths and Nested Xrefs

The saved path for an xref can be an absolute (fully specified) 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

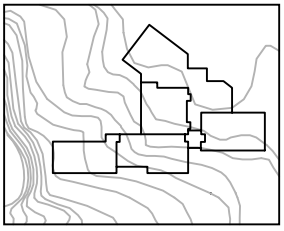
1. Click Insert menu ➤ External Reference.
2. In the Select Reference File dialog box, select the file you want to overlay. Click Open.
3. In the External Reference dialog box, under Reference Type, select Overlay.

   Using the Overlay option excludes any nested xrefs.
4. Specify insertion point, scale, and rotation angle, or click Specify On-Screen to use the pointing device.
5. Click OK.

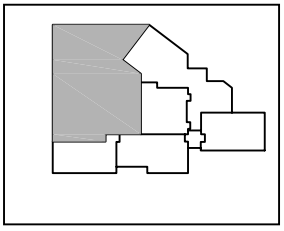
Reference toolbar

# Update Attached External References

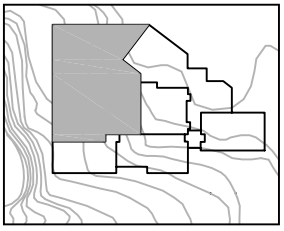
When you open a drawing, all xrefs update automatically. Use the  Reload option to update xrefs whenever you want to ensure that the most current versions are displayed in your drawing.



xref attached



edited xref file



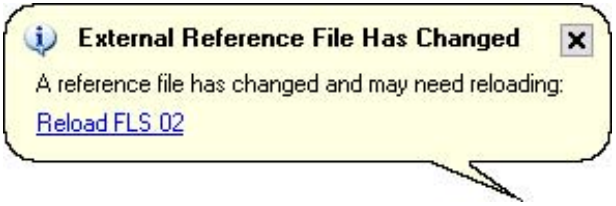
xref updated

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.

### 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  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 Xref Manager 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 shown.

---

#### Update Xrefs with Demand Loading Turned On

If demand loading is turned on when you load or reload an xref

- With the  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 Xrefs" on page 992.

#### To update an attached xref

1. Click Insert menu ➤ Xref Manager.
2. In the Xref Manager, select the reference name that you want to reload.
3. Select Reload. Click OK.

---

**NOTE** If the drawing you selected has been changed since you opened your drawing, the xref is reloaded.

---

Reference toolbar

# Clip External References and Blocks

After attaching a drawing as an xref or inserting a block, you can define a clipping boundary by using . A clipping boundary can define a portion of a block or xref while suppressing the display of geometry outside of the

boundary. Clipping applies to an individual instance of an xref, not to the xref definition itself. The portion of the xref or block within the clipped boundary remains visible, and the remainder of the xref or block becomes invisible. The referenced geometry is not altered; only the display of the xref is edited.

You can use XCLIP to create a new clipping boundary, delete an existing boundary, or generate a polyline object coincident with the vertices of the clipping boundary. Xref clipping can be turned on or off. When a clipping boundary is turned off, the boundary is not displayed and the entire xref is visible, provided that the geometry is on a layer that is on and thawed. When a clipping boundary is turned off, it still exists and can be turned on. However, deleting a clipping boundary is permanent.

After an xref or block has been clipped, it can be edited, moved, or copied just like an unclipped xref or block. The 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.

If you want to see the clipping boundary, you can turn on the  system variable. XCLIPFRAME determines whether the clipping boundary frame is displayed. When the clipping frame is on (set to 1), it can be selected as part of the object and plotted.

### Clipping Boundary Options

An xref clipping boundary can be specified as a rectangular window or a polygonal boundary. You can also select a polyline to define the clipping boundary. The 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.

### Rectangular Window Clipping Boundary

When you specify a rectangular clipping boundary, you are prompted for the corners of the window. The window is drawn parallel to the current UCS, and the clipping boundary is applied normal to the plane it lies on.

### Polygonal Clipping Boundary

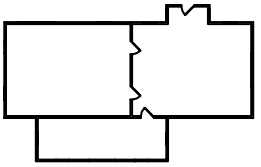
When you specify a polygonal clipping boundary, you are prompted to enter points to define the boundary. As you specify the clipping points, the last segment of the polygon is sketched so that the boundary is closed at all times. When you apply polygonal clipping to images in externally referenced drawings, the clipping boundary is applied to the rectangular extents of the polygonal boundary, rather than to the polygon itself.

### Polyline Clipping Boundary

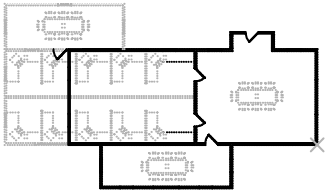
When you specify a polyline clipping boundary, you are prompted to select a 2D polyline object. The clipping boundary is created coincident with the polyline. Valid boundaries are 2D polylines with straight or spline-curved segments. Polylines with arc segments, or fit-curved polylines, can be used as the definition of the clip boundary, but the clip boundary will be created as a straight segment representation of that polyline. If the polyline has arcs, the clip boundary is created as if it had been decurved prior to being used as a clip boundary. An open polyline is treated as if it were closed.

### Clip Depth

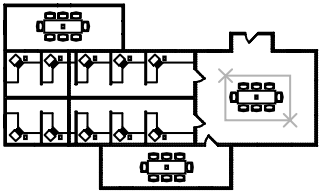
The front and back clipping planes for xrefs can be set. The xref must contain a clipping boundary before you can specify a *clip depth*. The clip depth is always calculated normal to the clipping boundary. When defining clip depth, you are prompted to specify a front and back point or distance relative to the clipped plane. Regardless of the current UCS, the clip depth is applied parallel to the clipping boundary.



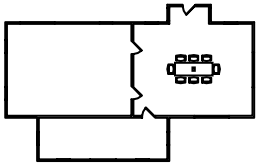
existing drawings



another similar drawing is referenced



xref is clipped to show only the conference table



resulting clipped xref

### To clip a reference

1 On the Command line, enter **xclip**.

2. Select a reference.
3. At the prompt, select New Boundary by pressing ENTER .
4. Select a rectangular or polygonal clipping boundary. Specify the corners or vertices of the boundary.

   The image is clipped based on the area that you specified and hides the portion of the xref outside the clipping boundary.

# 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 externally 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 an externally 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 remved from the name and replaces it 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 is bound and all of its dependent definitions.

**To bind xref-dependent named objects to the current drawing**

1. Click Modify menu ➤ Object ➤ External Reference ➤ Bind.
2. In the Xbind dialog box, click the plus (+) sign next to an external reference.

   The five types of named object definitions (Block, Dimstyle, Layer, Linetype, and Textstyle) are listed.
3. Click the plus (+) sign of one of the definition types.

   The names of the definition table entries are listed.
4. Select a named object definition. Click Add.

   The named object definition is listed under Definitions to Bind.
5. If necessary, repeat steps 3 and 4.
6. Click OK.

**To change the names of layers, dimension styles, and other named objects**

1. Click Format menu ➤ 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.

# Archive Drawings That Contain External References (Bind)

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 Bind option. For more information, see "Resolve Name Conflicts in External References" on page 973.

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".

---

**To bind an xref to the current drawing**

1. Click Insert menu ➤ Xref Manager.
2. In the Xref Manager, select an external reference. Click Bind.
3. 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.
4. Click OK to close each dialog box.

Reference toolbar

# Detach External References

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 Insert menu ➤ Xref Manager.
2. In the Xref Manager, select an external reference. Select Detach.
3. Click OK.

Reference toolbar

# Edit External References and Blocks in Place

There are two methods for editing xrefs: you can open the referenced drawing, 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 an Xref in a Separate Window

The simplest and most direct method for editing xrefs is to open the referenced drawing file in a separate window. This method gives you access to all objects in the referenced drawing.

Instead of browsing for the xref using the Select File dialog box, you can select the xref and open the referenced drawing. To use this method to edit xrefs, use the Xref Manager dialog box or the  command.

**To edit an xref in a separate window**

1. On the command line, enter **xopen**.
2. Select an object in an external reference.
3. In the new window, edit the referenced drawing file, save it, and then close window.

# Edit Xrefs and Blocks Within Context

By editing a reference in place, you can modify the reference within the visual context of your current drawing.

## Edit Selected Objects in Xrefs 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, and update the block definition. You cannot edit a block reference that is inserted using .

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.

---

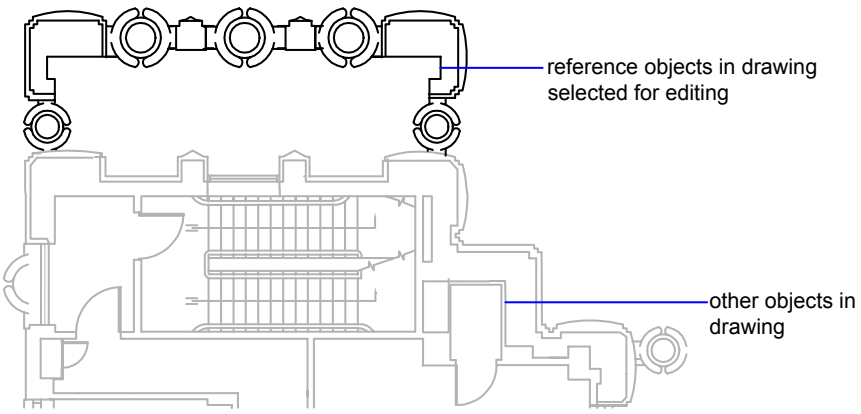
#### Understand the Working Set

Only the objects that are a part of the selected reference are available for selection in the drawing area. The objects you select from the selected reference are extracted and made available for modification in the current drawing. The set of extracted objects, or the *working set*, 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  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  is set to a value of 2D wireframe.

### Use the Refedit Toolbar

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.

### To edit an xref or block reference in place

1. Click Tools menu ➤ Xref and Block In-Place Editing ➤ Edit Reference In-Place.
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.

## Use the Working Set to Edit Xrefs and Blocks

To edit an external reference from within the current drawing, you use the *working set* to identify objects that belong to the external reference 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.

If a new object is created because of changes made to objects outside the working set, the new object is not added to the working set. For example, your drawing contains two lines that are not a part of the working set. If you edit the lines by using , a new arc is created between the two lines. The arc is not added to the working set.

When a reference object is part of the working set, you can select the object for editing even if it is drawn on a locked layer in the reference file. You can unlock the object's layer and make changes to the object. Changes made to the object can be saved, but the layer state remains the same in the reference file, whether it is locked or unlocked.

An object that is removed from the working set is added to the host drawing and removed from the reference when changes are saved back. An object that is added to the working set is removed from the host drawing and is 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  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.

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, or at the Command prompt, enter **refset**.
2. If you entered **refset** at the Command prompt, click the Add Objects to the Working Set button on the Refedit toolbar.

3. Select the objects you want to add. You can also set  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, or at the Command prompt, enter **refset**.
2. If you entered **refset** at the Command prompt, click the Remove Objects from the Working Set button on the Refedit toolbar.

3. Select the objects you want to remove. You can also set  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.

## Save Back Edited Xrefs and Blocks

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 . If you make unwanted changes to an xref and use  to save back the changes, you must use UNDO to undo any changes made during the reference editing session. After you have undone any unwanted changes, use REFCLOSE to save changes to restore the xref file to its original state.

---

**WARNING** While editing a reference in place, if you delete an object that is not in the working set, the object is not restored if you discard changes at the closing of the reference editing session.

---

Objects in the current drawing that inherit properties defined by the xref retain those new properties. Properties taken from the xref drawing are bound to the current drawing. The xref layer named SITE, for example, appears in the current drawing as $#$SITE when assigned to an object not in the working set. If  is set to 0, a prefix of $#$ is added to the reference name in the current drawing. If BINDTYPE is set to 1, reference names remain unchanged in the current drawing, similar to names of inserted objects.

---

**NOTE** When you edit and save an xref in place, the original drawing preview is no longer available unless you open and save the referenced drawing.

---

**To save back changes from edited xrefs and blocks**

- Click Tools menu ➤ Xref and Block In-Place Editing ➤ Save Reference Edits or click the Save Back Changes to Reference button on the Refedit toolbar.

Refedit toolbar

With no objects selected during in-place reference editing, right-click in the drawing area. Click Close Refedit Session.

**To discard all changes from edited xrefs and blocks**

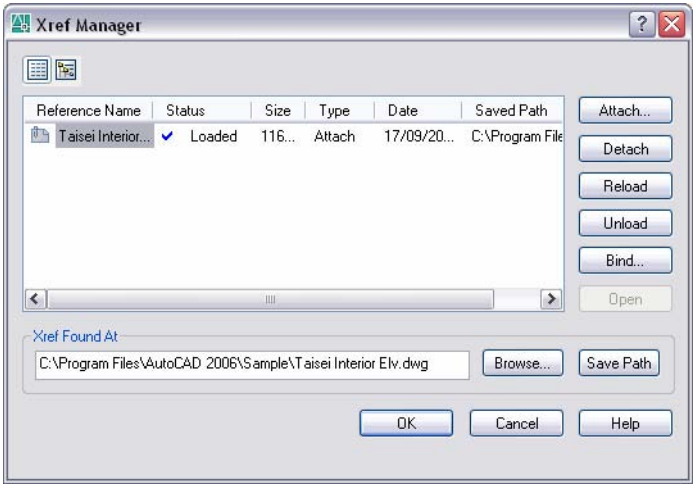
- On the Refedit toolbar, click the Discard Changes to Reference button.

Refedit toolbar

## Edit Xrefs and Blocks with Nesting, OLE, or Attributes

There are limitations with editing external references and blocks from your current drawing if they contain nested objects.

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.

# Set Paths to Externally Referenced Drawings

You can view and edit the file name and path that the program uses when loading an xref. Use this option if the xref drawing 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 external reference: an absolute path, a relative path, and no path.

### Specify an Absolute Path

An absolute path is a fully specified hierarchy of folders that locates the external reference. An absolute path includes the local hard drive letter or the 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 drawing 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 xrefs is moved or saved to a different path, to a different local hard drive, or to a different network server, you must edit any relative paths to accommodate the host drawing's new location or you must relocate the referenced files.

---

#### Specify No Path

When no path information is saved with the attached external reference, the following search is initiated in the order shown:

- Current folder of the host drawing
- Project search paths defined on the Files tab in the Options dialog box and in the  system variable
- Support search paths defined on the Files tab in the Options dialog box

■ Start In folder specified in the Microsoft® Windows® application shortcut

Specifying the No Path option is useful when moving a set of drawings to a different folder hierarchy or to an unknown folder hierarchy.

#### Know When an External Reference Has Been Relocated

If the drawing you are working on contains an xref that has been moved to a different folder, a message is displayed at the site of the xref when you load the drawing. The message indicates that the xref cannot be loaded using the old path. When you specify the new path, the xref is reloaded into your drawing.

#### Use Project Names to Define External Reference Paths

Project names make it easier for you to manage xrefs when drawings are exchanged between customers or if you have different drive mappings to the same location on a server. The project name points to a section in the registry that can contain one or more search paths for each project name defined.

If the program cannot find an xref at the location specified by the search path, the prefix (if any) is stripped from the path. If the drawing has a PROJECTNAME value set and a corresponding entry exists in the registry, the program searches for the file along the project search paths. If the xref still is not located, the program search path is searched again.

You can add, remove, or modify the project names that exist in the registry. The folder search paths beneath the project name can also be added, removed, or modified.

The search paths beneath the project name can be added, removed, or modified in the same manner as the project name. The order in which the folders are searched can also be modified. Projects and their search paths can only be edited through the Files tab in the Options dialog box. You cannot edit project names on the command line.

Once you have established a project name and the search paths you want associated with that project name, you can make that project name the currently active project. The program searches the paths associated with that currently active project for xrefs that were not found in the full search path, the current drawing folder, or the program support paths.

#### To change an xref path

1. Click Insert menu ➤ Xref Manager.
2. In the Xref Manager, select a reference name.

3 Under Xref Found At, do one of the following:

- Edit the xref path directly.
- Click Browse. 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.

Reference toolbar

**To display the currently defined project names**

1 Click Tools menu ➤ Options.

2 In the Options dialog box, Files tab, double-click Project Files Search Path.

3 Click each project name folder to display the search paths associated with it.

4 Click OK (or Apply).

**To add a project name**

1 Click Tools menu ➤ Options.

2 In the Options dialog box, Files tab, double-click Project Files Search Path. Click Add.

A folder named *projectx* (where *x* indicates the next available number) is created and indented beneath the project folder.

3 Either enter a new name, or press ENTER to accept *projectx*.

The project name must be 31 characters or fewer, and it cannot contain leading spaces or terminating spaces.

4 Click OK (or Apply).

**To remove a project name**

1 Click Tools menu ➤ Options.

2 In the Options dialog box, Files tab, double-click Project Files Search Path.

3 Select a project name. Click Remove.

4 Click OK (or Apply).

### To modify a project name

1 Click Tools menu ➤ Options.

2 In the Options dialog box, Files tab, select a project name. Enter a new name.

3 Click OK (or Apply).

You can also modify a project name by selecting the name in the project folder and pressing F2.

### To add a search path

1 Click Tools menu ➤ Options.

2 In the Options dialog box, Files tab, select a project name. Click Add.

3 Add a new search path beneath the project name by entering a new path, or click Browse and select a new path.

4 Click OK (or Apply).

The new path is indented and placed beneath the project name.

### To delete a search path

1 Click Tools menu ➤ Options.

2 In the Options dialog box, Files tab, select a project name. Click Remove.

3 Click OK.

### To change a search path

1 Click Tools menu ➤ Options.

2 In the Options dialog box, Files tab, select a project name. Click Browse.

3 In the Browse for Folder dialog box, select a new path.

4 Click OK to close each dialog box.

You can also change a search path by selecting the project path and pressing F2.

**To make a project current**

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Files tab, double-click Project Files Search Path.
3. Select a Project name. Click Set Current.
4. Click OK (or Apply).

   You can also set a project current by entering **projectname** on the command line and then entering the name of the project.

**To clear the current project**

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Files tab, click Project Files Search Path.
3. Click Clear Current.

   This clears the setting for the current drawing.
4. Click OK (or Apply).

   You can also clear the current project at the Command prompt by entering **projectname** and then entering a period (**.**).

# Resolve External Reference Errors

If an external reference cannot be loaded when you open a drawing, an error message is displayed.

## Resolve Missing External Reference Files

If an external reference 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 external reference. Each time you load or plot the drawing or use the xref Reload option in the Xref Manager to update the external reference, 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 external reference.

If the program cannot load an external reference when it is loading your drawing, it displays an error message. In this example, the program can't find the xref HOUSE:

"\acad\dwg\house.dwg": Can't open file

** Error resolving Xref HOUSE.

For each insertion of the external reference, 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  Path option to update or correct the path.

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
- Project search paths defined on the Files tab in the Options dialog box and in the  system variable
- Support search paths defined on the Files tab in the Options dialog box
- Start In folder specified in the Microsoft® Windows® application shortcut

This 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 Attached External References" on page 969

### To change an xref path

1. Click Insert menu ➤ Xref Manager.
2. In the Xref Manager dialog box, select an external reference.
3. Under Xref Found At, do one of the following:
   - Edit the xref path directly.
   - Click Browse. 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.

Reference toolbar

# Resolve External References That Are Circular

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 the program encounters a circular reference while loading a drawing, it displays an error message and breaks the circular reference for the current session. 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.

# Track External Reference Operations (Log File)

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  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
```

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.

# Increase Performance with Large Xrefs

There are several features that can improve performance with large external references.

## Overview of Demand Loading

Provides a high-level description of using demand loading to improve performance when working with large external references.

The program uses *demand loading* and saving drawings with indexes to increase performance with large external references 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 , , and system variables.

## Unload Xrefs

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  (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 Insert menu ➤ Xref Manager.
2. In the Xref Manager, select an xref. Click Unload.
3. Click OK.

Reference toolbar

# 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, and a spatial index is saved in the externally referenced drawing.
- Freeze several layers of the xref, and 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 external reference 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 externally referenced file 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.

## 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.

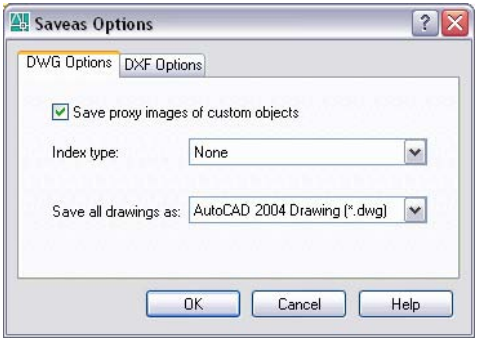
A layer index is a list showing which objects are on which layers. This list is used when the program is referencing the drawing in conjunction with demand loading to determine which objects need to be read in and displayed. Objects on frozen layers in an external reference are not read in if the external reference 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 external reference. 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 will not benefit from layer and spatial indexing or demand loading.

**To save a drawing with layer and spatial indexes**

1. Click File menu ➤ Save As.
2. In the Save Drawing As dialog box, click Tools menu ➤ Options.




3. In the Saveas Options dialog box, under Index Type, select Layer, Spatial, or Layer & Spatial. Click OK.

   **NOTE** If you are saving a partially open drawing that does not already contain layer and spatial indexes, this option is disabled.

4. Click Save.

# Set Paths for Temporary Xref File Copies

When you turn on demand loading with copy, the  system variable can be used to indicate the path where copies of externally referenced files 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  to 2 so that the externally referenced files are demand loaded from your local machine. Conversely, to minimize the number of temporary files created by multiple users referencing the same drawing, those users can set XLOADPATH to point to a common folder. In this manner, multiple sessions of the program can share the same temporary copies of reference drawings.

You can set XLOADPATH in the Options dialog box and indicate the path where copies of externally referenced files are to be placed.

**To set the file path for xref copies**

1. Click Tools menu ➤ Options.
2. In the Options dialog box, Files tab, select the Temporary External Reference File Location folder.

   The expanded tree view displays the path where copies of xref files are placed. If no path is specified, the temporary copies are placed in the location specified by Temporary Drawing File Location.
3. Edit the path by selecting it and entering a new path.
4. Click OK (or Apply) to set the path.

# Link and Embed Data (OLE)

# 30

Object linking and embedding (OLE) is a Windows feature that combines data from different applications into one document. For example, you can create an Adobe PageMaker layout that contains an AutoCAD drawing, or you can create an AutoCAD drawing that contains all or part of a Microsoft Excel spreadsheet.

## In this chapter


- Overview of Object Linking and Embedding
- Import OLE Objects
- Export OLE Objects from Drawings
- Edit OLE Objects in Drawings

# Overview of Object Linking and Embedding

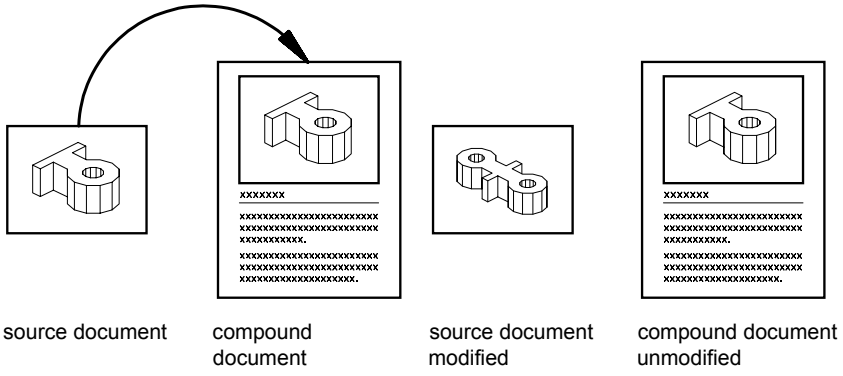
Object linking and embedding is a way to use information from one application in another application. To use OLE, you need both source and destination applications that support OLE.

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

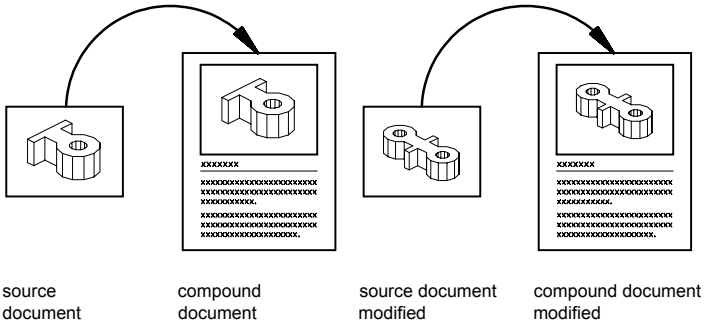
An embedded OLE object is a copy of information from another document. When you embed objects, there is no link to the source document and any changes made to the source document are not reflected in destination documents. Embed objects if you want to be able to use the application that created them 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  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  system variable.

---

**See also:**

"Use Windows Cut, Copy, and Paste" on page 594

**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.

# 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  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.

# Link OLE Objects in Drawings

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  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 Edit menu ➤ Paste Special.
5. In the Paste Special dialog box, click Paste Link.

   Paste Link pastes the contents of the Clipboard into the current drawing and creates a link to the file in the source application. If you click Paste, the Clipboard contents are embedded instead of linked.
6. In the As box, select the data format you want to use.
7. Click OK.

**To link an entire file as an OLE object in a drawing**

1. Open the drawing.
2. Click Insert menu ➤ 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. On the Edit menu, click OLE Links.
3. In the OLE Links dialog box, click Break Link.
4. Click Yes.

5 Click Close.

# 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 Edit menu ➤ Paste.
5. Click OK.

### To create an object in another application and embed it in a drawing

1. Open the drawing.
2. Click Insert menu ➤ 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.

# 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  command copies the view in the current viewport to the Clipboard. You can then paste the view into the destination document.

If you paste an unnamed view into a document, it is assigned a view name such as OLE1. If you then exit the drawing, you are prompted to save your changes to the newly named view. To establish the link and to save the view name, OLE1, you must save the drawing.

### Embed Drawing Objects in Other Documents

You can select objects and embed them in documents created by other applications. Embedding places a copy of the selected objects in the destination document. If you use AutoCAD to edit the OLE object from within the destination document, the object is not updated in the original drawing.

#### To embed objects in another document

1. Click Edit menu ➤ Copy.
2. In the drawing area, select the objects that you want to embed and then press  ENTER  .

   The selected objects are copied to the Clipboard.
3. Start the destination application, and open a new or existing document.
4. Paste the Clipboard contents into the document, following the destination application's instructions for embedding the Clipboard contents.

#### To link a view to another document

1. Save the drawing that you want to link, so that it has a drawing name.
2. If multiple viewports are displayed, select a viewport.
3. Click Edit menu ➤ Copy Link.

   The view is copied to the Clipboard.
4. Open a new or existing document in the destination application.
5. Paste the Clipboard contents into the document, following that application's procedures for inserting linked data.

   The inserted OLE object is displayed in the document and can be edited from AutoCAD through the destination application.

# 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: , , , and .

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.

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  system variable.

### Edit Information in OLE Objects

You can edit the information in linked or embedded OLE objects by double-clicking the object to open the source application.

### Edit OLE Objects When AutoCAD Is the Source Application

The document that contains a linked drawing stores the drawing's file location. You can edit a linked drawing either from the destination application or in the source program. The program must be loaded or accessible on the system along with the document you are editing.

An AutoCAD drawing that is embedded in a document can be edited only from within the destination application. Double-click the OLE object to start the program. Editing the original drawing in the program has no effect on documents in which that drawing is embedded.

**See also:**

"Copy Objects" on page 603

#### 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 Click File menu ➤ Save to save the changes to the drawing.

5 Click File menu ➤ Exit to return to the destination application.

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 Click File menu ➤ Save to save the changes to the drawing.

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 AutoCAD objects**

1 Open the document that contains the embedded AutoCAD objects (for example, a Microsoft Word file).

2 Double-click the embedded objects to start the program and display the objects.

3 Modify the objects as necessary.

4 Click File menu ➤ Update to save changes to the embedded objects.

5 Click File menu ➤ Exit to return to the destination application.

**To restore an OLE object to its original size and shape**

1 Select the OLE object.

2 Right-click. Click OLE ➤ Reset.

The Reset option is also available in the Text Size dialog box.

### To control the display of OLE objects

1. At the Command prompt, enter **olehide.**
2. Enter one of the following values:
   - **0** Displays OLE objects in both paper space and model space.
   - **1** Displays OLE objects in paper space only.
   - **2** Displays OLE objects in model space only.
   - **3** Does not display OLE objects.

### To turn the display of the frames of OLE objects on or off

1. At the Command prompt, enter **oleframe.**
2. Enter one of the following values:
   - **0** Frame is not displayed and not plotted.
   - **1** Frame is displayed and is plotted.
   - **2** Frame is displayed but is not plotted.

   The frame must be displayed in order for grips to be visible.

# Glossary

Commands associated with definitions are shown in parentheses at the end of the definition.

*X,Y,Z* point filters

*See* coordinate filters.

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

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)

activate

Part of the Autodesk software registration process. It allows you to run a product in compliance with the product's end-user license agreement.

adaptive sampling

A method to accelerate the anti-aliasing process within the bounds of the sample matrix size. *See also* anti-aliasing.

affine calibration

A tablet calibration method that provides an arbitrary linear transformation in two-dimensional space. Affine calibration requires three calibration points to allow a tablet transformation that combines translation, independent *X* and *Y* scaling, rotation, and some skewing. Use affine calibration if a drawing has been stretched differently in the horizontal or vertical direction. (TABLET)

alias

A shortcut for a command. For example, *CP* is an alias for COPY, and *Z* is an alias for ZOOM. You define aliases in the *acad.pgpaclt.pgp* file.

aliasing

The effect of discrete picture elements, or pixels, aligned as a straight or curved edge on a fixed grid appearing to be jagged or stepped. *See also* anti-aliasing.

aligned dimension

A dimension that measures the distance between two points at any angle. The dimension line is parallel to the line connecting the dimension's definition points. (DIMALIGNED)

ambient color

A color produced only by ambient light.

ambient light

Light that illuminates all surfaces of a model with equal intensity. Ambient light has no single source or direction and does not diminish in intensity over distance.

angular dimension

A dimension that measures angles or arc segments and consists of text, extension lines, and leaders. (DIMANGULAR)

angular unit

The unit of measurement for an angle. Angular units can be measured in decimal degrees, degrees/minutes/seconds, grads, and radians.

annotations

Text, dimensions, tolerances, symbols, or notes.

anonymous block

An unnamed block created by a number of features, including associative and nonassociative dimensions.

ANSI

For *American National Standards Institute*. Coordinator of voluntary standards development for both private and public sectors in the United States. Standards pertain to programming languages, Electronic Data Interchange

(EDI), telecommunications, and the physical properties of diskettes, cartridges, and magnetic tapes.

anti-aliasing

A method that reduces aliasing by shading the pixels adjacent to the main pixels that define a line or boundary. *See also* "aliasing" on page 1015.

approximation points

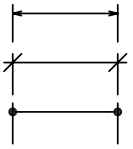
Point locations that a B-spline must pass near, within a fit tolerance. *See also* fit points *and* interpolation points.

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.




ASCII

For *American Standard Code for Information Interchange*. A common numeric code used in computer data communications. The code assigns meaning to 128 numbers, using seven bits per character with the eighth bit used for parity checking. Nonstandard versions of ASCII assign meaning to 255 numbers.

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 " on page 1026.

associative hatch

Hatching that conforms to its bounding objects such that modifying the bounding objects automatically adjusts the hatch. (BHATCH)

attenuation

The diminishing of light intensity over distance.

attribute definition

An object that is included in a block definition to store alphanumeric data. Attribute values can be predefined or specified when the block is inserted. Attribute data can be extracted from a drawing and inserted into external files. (ATTDEF)

attribute extraction file

An ASCII 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

An ASCII 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" on page 1017.

attribute prompt

The text string displayed when you insert a block with an attribute whose value is undefined. *See also* attribute definition, attribute tag, *and* attribute value.

attribute tag

A text string associated with an attribute that identifies a particular attribute during extraction from the drawing database. *See also attribute definition*, attribute prompt , *and* attribute value.

attribute value

The alphanumeric information associated with an attribute tag. *See also attribute definition*, attribute prompt, *and attribute tag*.

AutoCAD library search path

The order in which looks for a support file: current directory, drawing directory, directory specified in the support path, and directory containing the executable file, *acad.exe*.

AutoCAD window

The drawing area, its surrounding menus, and the command line.

axis tripod

Icon with *X*, *Y*, and *Z* coordinates that is used to visualize the viewpoint (view direction) of a drawing without displaying the drawing. (VPOINT)

B-spline curve

A blended piecewise polynomial curve passing near a given set of control points. *See also* Bezier curve. (SPLINE)

back face

The opposite side of a front face. Back faces are not visible in a rendered image. *See also* front faces.

base point

1. In the context of editing grips, the grip that changes to a solid color when selected to specify the focus of the subsequent editing operation. 2. A point for relative distance and angle when copying, moving, and rotating objects. 3. The insertion base point of the current drawing. (BASE) 4. The insertion base point for a block definition. (BLOCK)

baseline

An imaginary line on which text characters appear to rest. Individual characters can have descenders that drop below the baseline. *See also* baseline dimension.

baseline dimension

Multiple dimensions measured from the same baseline. Also called *parallel dimensions*. *See also* baseline.

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 authoring palettes

Tool palettes used in the Block Editor to add actions and parameters to dynamic block definitions.

block authoring tools

Actions, parameters, and parameter sets on the tabs of the Block Authoring Palettes window. Used in the Block Editor to create dynamic blocks.

block 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 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)

bump map

A map in which brightness values are translated into apparent changes in the height of the surface of an object.

button menu

The menu for a pointing device with multiple buttons. Each button on the pointing device (except the pick button) can be defined in the customization file (*acad.cui*).

BYBLOCK

A special object property used to specify that the object inherits the color or linetype of any block containing it. *See also* BYLAYER.

BYLAYER

A special object property used to specify that the object inherits the color or linetype associated with its layer. *See also* BYBLOCK.

callout block

A block used as symbol to reference another sheet. Callout blocks have many industry-specific terms, such as reference tags, detail keys, detail markers, and so on. *See also* label block.

category

*See* view category.

Chain Actions

In a dynamic block definition, a property of point, linear, polar, XY, and rotation parameters. When set to Yes, a change in an action that contains the parameter in the action's selection set triggers any actions associated with that parameter, just as if you had edited the parameter in the block reference through a grip or custom property.

circular external reference

An externally referenced drawing (xref) that references itself directly or indirectly. The xref that creates the circular condition is ignored.

clipping planes

The boundaries that define or clip the field of view. (DVIEW)

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 map

A table defining the intensity of red, green, and blue (RGB) for each displayed color.

command line

A text area reserved for keyboard input, prompts, and messages.

construction plane

A plane on which planar geometry is constructed. The *XY* plane of the current UCS represents the construction plane. *See also* elevation *and* user coordinate system (UCS).

continued dimension

A type of linear dimension that uses the second extension line origin of a selected dimension as its first extension line origin, breaking one long dimension into shorter segments that add up to the total measurement. Also called *chain dimension*. (DIMCONTINUE)

control frame

A series of point locations used as a mechanism to control the shape of a B-spline. These points are connected by a series of line segments for visual clarity and to distinguish the control frame from fit points. The SPLFRAME system variable must be turned on to display control frames.

control point

*See* control frame.

Coons patch

In 3D surface meshes, the bicubic surface (one curved in the M direction and another in the N direction) interpolated between four edges.

coordinate filters

Functions that extract individual *X*, *Y*, and *Z* coordinate values from different points to create a new, composite point. Also called *X,Y,Z point filters*.

crosshairs

A type of cursor consisting of two lines that intersect.

crossing selection

A rectangular area drawn to select objects fully or partly within its borders.

CTB file

*S*A color-dependent plot style table.

cursor

*See* pointer and crosshairs.

cursor menu

*See* shortcut menu.

custom grips

In a dynamic block reference, used to manipulate the geometry and custom properties.

custom object

A type of object that is created by an ObjectARX application and that typically has more specialized capabilities than standard objects. Custom objects include parametric solids (Autodesk Mechanical Desktop), intelligently interactive door symbols (Autodesk Architectural Desktop), polygon objects (Autodesk Map), 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 releases prior to AutoCAD 2006.

default

A predefined value for a program input or parameter. Default values and options for commands are denoted by angle brackets (<>).

default drawing

*See* initial environment.

definition points

Points for creating a dimension. The program refers to the points to modify the appearance and value of a nonassociative dimension when the

dimensioned object is modified. Also called *defpoints* and stored on the special layer DEFPOINTS.

definition table

The nongraphical data area of a drawing file that stores block definitions.

dependency highlighting

In a dynamic block definition, how associated objects are displayed when you select a parameter, grip, or action.

dependent named objects (**in xrefs**)

Named objects brought into a drawing by an external reference. *See also* named object *and* symbol table**.**

dependent symbols

*See* dependent named objects (in xrefs).

DIESEL

For *Direct Interpretively Evaluated String Expression Language*. A macro language for altering the status line with the MODEMACRO system variable and for customizing menu items.

diffuse color

An object's predominant color.

dimension line arc

An arc (usually with arrows at each end) spanning the angle formed by the extension lines of an angle being measured. The dimension text near this arc sometimes divides it into two arcs. *See also* angular dimension.

dimension style

A named group of dimension settings that determines the appearance of the dimension and simplifies the setting of dimension system variables. (DIMSTYLE)

dimension text

The measurement value of dimensioned objects.

dimension variables

A set of numeric values, text strings, and settings that control dimensioning features. (DIMSTYLE)

direct distance entry

A method to specify a second point by first moving the cursor to indicate direction and then entering a distance.

dithering

Combining color dots to give the impression of displaying more colors than are actually available.

dockable window

A user interface element that can be either docked or floating in the drawing area. Dockable windows include the command window, tool palettes, Properties Palette, and so on.

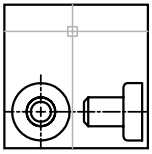
drawing area

drawing area

The area in which your drawings are displayed and modified. The size of the drawing area varies, depending on the size of the AutoCAD window and on how many toolbars and other elements are displayed. *See also* AutoCAD window.

drawing extents

The smallest rectangle that contains all objects in a drawing, positioned on the screen to display the largest possible view of all objects. (ZOOM)



drawing extents

drawing limits

*See* grid limits.

drawing set

A collection of drawings assembled using the Publish dialog box.

drawing template

A drawing file with preestablished settings for new drawings such as *acad.dwt, aclt.dwt* and *acadiso.dwt, acltiso.dwt;* however, any drawing can be used as a template. *See also* initial environment.

DSD

For *drawing set descriptions*. A file format for saving a description of a drawing set that has been assembled using the Publish dialog box.

DST

For *sheet set data*. The XML file format used to store the associations and information that define a sheet set.

DWF

For *Design Web Format*. A highly compressed file format that is created from a DWG file. DWF files are easy to publish and view on the Web. *See also* DWG *and* DXF.

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.

edge

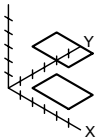
The boundary of a face.

electronic drawing set

The digital equivalent of a set of plotted drawings. You create an electronic drawing set by publishing drawings to a DWF file.

elevation

The default *Z* value above or below the *XY* plane of the current user coordinate system, which is used for entering coordinates and digitizing locations. (ELEV)



elevation

embed

To use object linking and embedding (OLE) information from a source document in a destination document. An embedded object is a copy of the information from a source document that is placed in the destination document and has no link to the source document. *See also* link.

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 variable

A setting stored in the operating system that controls the operation of a program.

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)

extents

*See* drawing extents.

external reference (xref)

A drawing file referenced by another drawing. (XREF)

extrusion

A 3D solid created by sweeping an object that encloses an area along a linear path.

face

A triangular or quadrilateral portion of a surface object.

feature control frame

The tolerance that applies to specific features or patterns of features. Feature control frames always contain at least a geometric characteristic symbol to indicate the type of control and a tolerance value to indicate the amount of acceptable variation.

fence

A multisegmented line specified to select objects it passes through.

field

A specialized text object set up to display data that may change during the life cycle of the drawing. When the field is updated, the latest value of the field is displayed. (FIELD)

fill

A solid color covering an area bounded by lines or curves. (FILL)

filters

*See* coordinate filters.

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 viewports

*See* layout viewports.

font

A character set, comprising letters, numbers, punctuation marks, and symbols of a distinctive proportion and design.

freeze

A setting that suppresses the display of objects on selected layers. Objects on frozen layers are not displayed, regenerated, or plotted. Freezing layers shortens regenerating time. *See also* thaw. (LAYER)

front faces

Faces with their normals pointed outward.

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 screen

*See* drawing area

graphics window

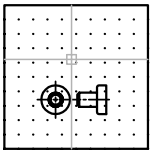
*See* AutoCAD window *and* drawing area.

grid

An area covered with regularly spaced dots to aid drawing. The spacing between grid dots is adjustable. Grid dots are not 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 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.

handle

A unique alphanumeric tag for an object in the program's database.

HDI

For *Heidi Device Interface*. An interface for developing device drivers that are required for peripherals to work with the program and other Autodesk products.

HLS

For *hue, lightness, and saturation*. A system of defining color by specifying the amount of hue, lightness, and saturation.

home page

The main navigating screen for a Web site.

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*. An ANSI-standard format for digital representation and exchange of information between CAD/CAM systems. *See also* ANSI.

initial environment

The variables and settings for new drawings as defined by the default drawing template, such as *acad.dwg, aclt.dwg* or *acadiso.dwg, acltiso.dwg*. *See also* template drawing.

input property

In a dynamic block definition, a parameter property other than that of a lookup, alignment, or base point parameter that you can add as a column to a lookup table. When the parameter values in a dynamic block reference match a row of input property values, the corresponding lookup property values in that table row are assigned to the block reference. (BLOOKUPTABLE)

interface element

A user interface object that can be customized, such as a toolbar, pull-down menu, shortcut key, dockable window, and so on.

interpolation points

Defining points that a B-spline passes through. *See also* approximation points *and* fit points.

island

An enclosed area within another enclosed area. Islands may be detected as part of the process of creating hatches, polylines, and regions. (BHATCH, BOUNDARY)

ISO

For *International Standards Organization*. The organization that sets international standards in all fields except electrical and electronics. Headquarters are in Geneva, Switzerland.

isometric snap style

A drafting option that aligns the cursor with two of three isometric axes and displays grid points, making isometric drawings easier to create.

key point

In a dynamic block definition, the point on a parameter that drives its associated action when edited in the block reference.

label block

A block used to label views and details. Labels contain data, such as a title, view number, and scale, that is associated with the referenced view. *See also* callout block.

layer

A logical grouping of data that are like transparent acetate overlays on a drawing. You can view layers individually or in combination. (LAYER)

layer index

A list showing the objects on each layer. A layer index is used to locate what portion of the drawing is read when you partially open a drawing. Saving a layer index with a drawing also enhances performance when you work with external references. The INDEXCTL system variable controls whether layer and spatial indexes are saved with a drawing.

layer translation mappings

Assignments of a set of layers to another set of layers that defines standards. These standards include layer names and layer properties. Also called *layer mappings*.

layout

The tabbed 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)

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.

lookup property

In a dynamic block definition, a lookup parameter that you add to a lookup table. The lookup parameter label is used as the property name. When the parameter values in a dynamic block reference match a row of input property values, the corresponding lookup property values in that table row are assigned to the block reference. (BLOOKUPTABLE)

lookup table

Defines properties for and assigns property values to a dynamic block. Assigns property values to the dynamic block reference based on how the block is manipulated in a drawing. (BLOOKUPTABLE)

main customization file

A writable CUI file that defines most of the user interface elements (including the standard menus, toolbars, keyboard accelerators, and so on). The *acad.cui* file (the default main CUI file) is automatically loaded when you start AutoCAD.

markup

A single comment or a redline geometry correction inserted into a DWF file using Autodesk DWF Composer.

markup set

A group of markups contained within a single DWF file.

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.

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 view

A view saved for restoration later. (VIEW)

node

An object snap specification to locate points, dimension definition points, and dimension text origins.

nonassociative dimension

A dimension that does not automatically change as the associated geometry is modified. Controlled by the DIMASSOC system variable. *See also* associative dimension *and* exploded dimension.

normal

A vector that is perpendicular to a face.

noun-verb selection

Selecting an object first and then performing an operation on it rather than entering a command first and then selecting the object.

NURBS

For *nonuniform rational B-spline curve*. A B-spline curve or surface defined by a series of weighted control points and one or more knot vectors. *See also* B-spline curve.

object

One or more graphical elements, such as text, dimensions, lines, circles, or polylines, treated as a single element for creation, manipulation, and modification. Formerly called *entity*.

object enabler

A tool that provides specific viewing and standard editing access to a custom object when the ObjectARX application that created the custom object is not present. *See also* custom object and proxy object.

Object Snap mode

Methods for selecting commonly needed points on an object while you create or edit a drawing. *See also* running object snap *and* object snap override.

object snap override

Turning off or changing a running Object Snap mode for input of a single point. *See also* Object Snap mode *and* running object snap.

ObjectARX (AutoCAD Runtime Extension)

A compiled-language programming environment for developing AutoCAD applications.

OLE

For *object linking and embedding*. An information-sharing method in which data from a source document can be linked to or embedded in a destination document. Selecting the data in the destination document opens the source application so that the data can be edited. *See also* embed *and* link.

opacity map

Projection of opaque and transparent areas onto objects, creating the effect of a solid surface with holes or gaps.

origin

The point where coordinate axes intersect. For example, the origin of a Cartesian coordinate system is where the *X*, *Y*, and *Z* axes meet at 0,0,0.

Ortho mode

A setting that limits pointing device input to horizontal or vertical (relative to the current snap angle and the user coordinate system). *See also* snap angle *and* user coordinate system (UCS).

orthogonal

Having perpendicular slopes or tangents at the point of intersection.

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 paper space viewports using a layout tab. Model space is used for creating the drawing. You design your model using the Model tab. *See also* model space *and* viewpoint. (PSPACE)

parameter

In a dynamic block definition, defines custom properties for the dynamic block by specifying positions, distances, and angles for geometry in the block.

parameter set

A tool on the Parameter Sets tab of the Block Authoring Palettes window that adds one or more parameters and one or more associated actions to the dynamic block definition.

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 *acad.exeaclt.exe* during installation, by entering the user name, company, and other information.

perspective view

Objects in 3D seen by an observer positioned at the viewpoint looking at the view center. Objects appear smaller when the distance from the observer (at the view point) to the view center increases. Although a perspective view appears realistic, it does not preserve the shapes of objects. Parallel lines seemingly converge in the view, so measurements cannot be made to scale from perspective views. The program has perspective view settings for VPORTS table entries as well as viewport objects. When you are looking at a viewport with a perspective view, the UCS icon has a different appearance.

photorealistic rendering

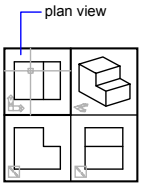
Rendering that resembles a photograph.

pick button

The button on a pointing device that is used to select objects or specify points on the screen. For example, on a two-button mouse, it is the left button.

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.

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)

procedural materials

Materials that generate a 3D pattern in two or more colors, and apply it to an object. These include marble, granite, and wood. Also called *template materials*.

prompt

A message on the command line 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.

PWT

A template file format used to publish drawings to the Web.

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)

reflection color

The color of a highlight on shiny material. Also called *specular color*.

reflection line

In a dynamic block reference, the axis about which a flip action's selection set flips when the associated parameter is edited through a grip or the Properties palette.

reflection mapping

Creates the effect of a scene reflected on the surface of a shiny object.

regenerate

To update a drawing's screen display by recomputing the screen coordinates from the database. *See also* redraw. (REGEN)

relative coordinates

Coordinates specified in relation to previous coordinates.

resource drawing

A drawing that is used as a data resource for the sheet set. You can place a saved model space view from a resource drawing onto a sheet.

return button

The button on a pointing device used to accept an entry. For example, on a two-button mouse, it is the right button.

reverse lookup

Adds a lookup grip to a dynamic block reference. When you click this grip, a drop-down list of the lookup values for that lookup property (column in the lookup table) is displayed. When you select a value from the list, the corresponding input property values are assigned to the block reference.

Depending on how the block was defined, this usually results in a change in the block reference's geometry. (BLOOKUPTABLE)

RGB

For *red, green, and blue*. A system of defining colors by specifying percentages of red, green, and blue.

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)

save back

To update the objects in the original reference (external or block reference) with changes made to objects in a working set during in-place reference editing.

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*.

selection set

One or more selected objects that a command can act upon at the same time.

In a dynamic block definition, the geometry associated with an action.

ShapeManager

The technology used to produce 3D solids.

sheet

A layout selected from a drawing file and assigned to a sheet set. *See also* sheet set.

sheet list table

A table listing all sheets in a sheet set. A sheet list table can be generated automatically with the Sheet Set Manager.

sheet selection

A named selection of sheets in a sheet set that can be conveniently recalled for archiving, transmitting, and publishing operations.

sheet set

An organized and named collection of sheets from several drawing files. *See also* sheet. (SHEETSET)

shortcut keys

Keys and key combinations that start commands; for example, CTRL + S saves a file. The function keys (F1, F2, and so on) are also shortcut keys. Also known as *accelerator keys*.

shortcut menu

The menu displayed at your cursor location when you right-click your pointing device. The shortcut menu and the options it provides depend on the pointer location and other conditions, such as whether an object is selected or a command is in progress.

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.

smooth shading

Smoothing of the edges between polygon faces.

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.

spatial index

A list that organizes objects based on their location in space. A spatial index is used to locate what portion of the drawing is read when you partially open a drawing. Saving a spatial index with a drawing also enhances performance when working with external references. The  system variable controls whether layer and spatial indexes are saved with a drawing.

specular reflection

The light in a narrow cone where the angle of the incoming beam equals the angle of the reflected beam.

STB file

For *plot style table* file. Contains plot styles and their characteristics.

stretch frame

In a dynamic block definition that contains a stretch action or a polar stretch action, determines how the objects within or crossed by the frame are edited in the block reference.

Subscription Center

A resource available from the Help menu for subscription members to access the latest releases of Autodesk software, incremental product enhancements, personalized web support, and self-paced e-Learning.

subset

A named collection of sheets in a sheet set that is often organized by discipline or workflow stage. *See also* view category.

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

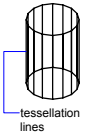
A rectangular array of cells that contain annotation, primarily text but also blocks. In the AEC industry, tables are often referred to as "schedules" and contain information about the materials needed for the construction of the building being designed. In the manufacturing industry, they are often referred to as "BOM" (bills of materials). (TABLE)

temporary files

Data files created during an program session. The files are deleted by the time you end the session. If the session ends abnormally, such as during a power outage, temporary files might be left on the disk.

tessellation lines

Lines that help you visualize a curved surface.




text style

A named, saved collection of settings that determines the appearance of text characters—for example, stretched, compressed, oblique, mirrored, or set in a vertical column.

texture map

The projection of an image (such as a tile pattern) onto an object (such as a chair).

thaw

A setting that displays previously frozen layers. *See also* freeze. (LAYER)

thickness

The distance certain objects are extruded to give them a 3D appearance. (PROPERTIES, CHPROP, ELEV, THICKNESS)

tiled viewports

*See* model viewports.

TILEMODE

A system variable that controls whether viewports can be created as movable, resizable objects (layout viewports), or as nonoverlapping display elements that appear side-by-side (model viewports). *See also* viewport.

toolbar

Part of the interface containing icons that represent commands.

tracking

A way to locate a point relative to other points on the drawing.

transparent command

A command started while another is in progress. Precede transparent commands with an apostrophe.

UCS

*See* user coordinate system (UCS).

UCS icon

An icon that indicates the orientation of the UCS axes. (UCSICON)

UCS icon

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).

value set

In a dynamic block definition, a range or list of values specified for a linear, polar, XY, or rotation parameter.

vector

A mathematical object with precise direction and length but without specific location.

vertex

A location where edges or polyline segments meet.

view

A graphical representation of a model from a specific location (viewpoint) in space. *See also* viewpoint *and* viewport. (VPOINT, DVIEW, VIEW)

view category

A named collection of views in a sheet set that is often organized by function. *See also* subset.

viewpoint

The location in 3D model space from which you are viewing a model. *See also* view *and* viewport. (DVIEW, VPOINT)

viewport

A bounded area that displays some portion of the model space of a drawing. The TILEMODE system variable determines the type of viewport created. 1. When TILEMODE is off (0), viewports are objects that can be moved and resized on a layout. (MVIEW) 2. When TILEMODE is on (1), the entire drawing area is divided into nonoverlapping model viewports. *See also* TILEMODE, view, *and* viewpoint. (VPORTS)

viewport configuration

A named collection of model viewports that can be saved and restored. (VPORTS)

virtual screen display

The area in which the program can pan and zoom without regenerating the drawing.

visibility mode

Displays or does not display geometry (in a dimmed state) that is invisible for a visibility state. (BVMODE)

visibility state

In a dynamic block, a custom property that allows only specified geometry to display in the block reference. (BVSTATE)

volumetric shadows

A photorealistically rendered volume of space cast by the shadow of an object.

WCS

*See* world coordinate system (WCS).

window selection

A rectangular area specified in the drawing area to select multiple objects at the same time. *See also* crossing selection, polygon window selection.

wipeout object

A polygonal area that masks underlying objects with the current background color. This area is bounded by the wipeout frame, which you can turn on for editing and turn off for plotting.

wireframe model

The representation of an object using lines and curves to represent its boundaries.

working drawing

A drawing for manufacturing or building purposes.

working set

A group of objects selected for in-place reference editing.

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).

WCS icon

xref

*See* external reference (xref).

zoom

To reduce or increase the apparent magnification of the drawing area. (ZOOM)

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