**Organize Drawings and Layouts** ............................................. 127

Chapter 13 **Create Single-View Drawings (Model Space)** .......... 129
  Quick Start for Model Space Drafting ....................................... 129
  Draw, Scale, and Annotate in Model Space .............................. 131

Chapter 14 **Create Multiple-View Drawing Layouts (Paper Space)** ... 137
  Quick Start for Layouts .......................................................... 137
  Understand the Layout Process ............................................... 139
  Work with Model Space and Paper Space ................................. 141
    Work on the Model Layout .................................................. 141
    Work on a Named Layout ................................................... 142
    Access Model Space from a Layout Viewport ......................... 144
  Create and Modify Layout Viewports ..................................... 145
  Control Views in Layout Viewports ....................................... 147
    Scale Views in Layout Viewports ........................................ 147
    Control Visibility in Layout Viewports ............................... 149
    Scale Linetypes in Layout Viewports .................................. 153
    Align Views in Layout Viewports ....................................... 154
    Rotate Views in Layout Viewports ..................................... 155
  Reuse Layouts and Layout Settings ....................................... 157

Create and Modify Objects .................................................. 159

Chapter 15 **Control the Properties of Objects** ....................... 161
  Work with Object Properties ................................................ 161
    Overview of Object Properties ......................................... 161
    Display and Change the Properties of Objects ........................ 163
    Copy Properties Between Objects ....................................... 164
  Work with Layers ............................................................... 165
    Overview of Layers ......................................................... 165
    Use Layers to Manage Complexity ..................................... 167
    Create and Name Layers ................................................... 169
    Change Layer Settings and Layer Properties .......................... 171
    Override Layer Properties in Viewports ............................... 173
  Work with Colors ............................................................... 176
    Set the Current Color ...................................................... 176
    Change the Color of an Object .......................................... 177
    Use Color Books .............................................................. 179
  Work with Linetypes .......................................................... 180
    Overview of Linetypes ..................................................... 180
    Load Linetypes ............................................................... 181
    Set the Current Linetype .................................................. 182
### Chapter 16 Use Precision Tools

- **Use Coordinates and Coordinate Systems (UCS)** .... 199
- **Overview of Coordinate Entry** .... 199
- **Enter 2D Coordinates** .... 200
- **Enter 3D Coordinates** .... 205
- **Understand the User Coordinate System (UCS)** .... 210
- **Specify Workplanes in 3D (UCS)** .... 214
- **Assign User Coordinate System Orientations to Viewports** .... 219
- **Control the Display of the User Coordinate System Icon** .... 221
- **Use Dynamic Input** .... 223
- **Snap to Locations on Objects (Object Snaps)** .... 226
- **Use Object Snaps** .... 226
- **The Object Snap Menu** .... 229
- **Set Visual Aids for Object Snaps (AutoSnap)** .... 229
- **Override Object Snap Settings** .... 230
- **Restrict Cursor Movement** .... 232
- **Adjust Grid and Grid Snap** .... 232
- **Use Orthogonal Locking (Ortho Mode)** .... 237
- **Use Polar Tracking and PolarSnap** .... 238
- **Lock an Angle for One Point (Angle)** .... 241
- **Combine or Offset Points and Coordinates** .... 241
- **Combine Coordinate Values (Coordinate Filters)** .... 241
- **Track to Points on Objects (Object Snap Tracking)** .... 244
- **Track to Offset Point Locations (Tracking)** .... 246
- **Specify Distances** .... 247
- **Enter Direct Distances** .... 247
- **Offset from Temporary Reference Points** .... 248
- **Specify Intervals on Objects** .... 248
- **Extract Geometric Information from Objects** .... 252
- **Obtain Distances, Angles, and Point Locations** .... 252
- **Obtain Area and Mass Properties Information** .... 253
Use a Calculator ............................................. 257
Use the Command Prompt Calculator .............. 257

**Chapter 17** Draw Geometric Objects .............. 261

- Draw Linear Objects ...................................... 261
- Draw Lines .................................................. 261
- Draw Polylines ............................................. 262
- Draw Rectangles and Polygons ......................... 265
- Draw Multiline Objects ................................... 267
- Draw Freehand Sketches .................................. 269

- Draw Curved Objects ...................................... 270
  - Draw Arcs ................................................ 270
  - Draw Circles ............................................ 274
  - Draw Polyline Arcs .................................... 275
  - Draw Donuts ............................................ 279
  - Draw Ellipses ........................................... 280
  - Draw Splines ............................................ 281
  - Draw Helixes ............................................ 285

- Draw Construction and Reference Geometry .......... 286
  - Draw Reference Points .................................. 286
  - Draw Construction Lines (and Rays) .................. 287

- Create and Combine Areas (Regions) .................. 289
- Create Revision Clouds .................................. 291

**Chapter 18** Change Existing Objects ............... 293

- Select Objects ............................................ 293
  - Select Objects Individually ............................... 293
  - Select Multiple Objects .................................. 295
  - Prevent Objects from Being Selected .................. 298
  - Select Objects by Properties ............................. 299
  - Customize Object Selection ............................... 300
  - Group Objects .......................................... 304

- Correct Mistakes ......................................... 307
- Erase Objects ............................................. 309
- Cut, Copy, and Paste with the Clipboard .............. 310
- Modify Objects ............................................ 311
  - Choose a Method to Modify Objects ....................... 311
  - Edit Objects with Grips .................................. 313
  - Move or Rotate Objects ................................... 321
  - Copy, Offset, or Mirror Objects ......................... 325
  - Change the Size and Shape of Objects ................. 337
  - Fillet, Chamfer, Break, or Join Objects .................. 344

- Modify Complex Objects .................................. 353
  - Disassociate Compound Objects (Explode) .............. 353
  - Modify Polylines ........................................ 355
<table>
<thead>
<tr>
<th>Chapter 22 Create 3D Models</th>
<th>441</th>
</tr>
</thead>
<tbody>
<tr>
<td>Overview of 3D Modeling</td>
<td>441</td>
</tr>
<tr>
<td>Create Solids and Surfaces from Lines and Curves</td>
<td>446</td>
</tr>
<tr>
<td>Overview of Creating Solids and Surfaces</td>
<td>446</td>
</tr>
<tr>
<td>Create a Solid or Surface by Extruding</td>
<td>450</td>
</tr>
<tr>
<td>Create a Solid or Surface by Sweeping</td>
<td>453</td>
</tr>
<tr>
<td>Create a Solid or Surface by Lofting</td>
<td>456</td>
</tr>
<tr>
<td>Create a Solid or Surface by Revolving</td>
<td>459</td>
</tr>
<tr>
<td>Create Solids</td>
<td>461</td>
</tr>
<tr>
<td>Overview of Creating 3D Solids</td>
<td>461</td>
</tr>
<tr>
<td>Create 3D Solid Primitives</td>
<td>465</td>
</tr>
<tr>
<td>Create a Polysolid</td>
<td>475</td>
</tr>
<tr>
<td>Create 3D Solids from Objects</td>
<td>477</td>
</tr>
<tr>
<td>Combine or Slice 3D Objects</td>
<td>481</td>
</tr>
<tr>
<td>Check 3D Models for Interferences</td>
<td>484</td>
</tr>
<tr>
<td>Create Surfaces</td>
<td>485</td>
</tr>
<tr>
<td>Overview of Creating Surfaces</td>
<td>486</td>
</tr>
<tr>
<td>Create Procedural Surfaces</td>
<td>491</td>
</tr>
<tr>
<td>Create NURBS Surfaces</td>
<td>501</td>
</tr>
<tr>
<td>Create Associative Surfaces</td>
<td>505</td>
</tr>
<tr>
<td>Create Meshes</td>
<td>511</td>
</tr>
<tr>
<td>Overview of Creating Meshes</td>
<td>511</td>
</tr>
<tr>
<td>Create 3D Mesh Primitives</td>
<td>515</td>
</tr>
<tr>
<td>Construct Meshes from Other Objects</td>
<td>528</td>
</tr>
<tr>
<td>Create Meshes by Conversion</td>
<td>534</td>
</tr>
<tr>
<td>Create Custom Mesh (Legacy)</td>
<td>537</td>
</tr>
<tr>
<td>Create Wireframe Models</td>
<td>542</td>
</tr>
<tr>
<td>Add 3D Thickness to Objects</td>
<td>544</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Chapter 23 Modify 3D Models</th>
<th>547</th>
</tr>
</thead>
<tbody>
<tr>
<td>Overview of Modifying 3D Objects</td>
<td>547</td>
</tr>
<tr>
<td>Use Gizmos to Modify Objects</td>
<td>549</td>
</tr>
<tr>
<td>Overview of Using Gizmos</td>
<td>549</td>
</tr>
<tr>
<td>Use the Gizmos</td>
<td>551</td>
</tr>
<tr>
<td>Move 3D Objects</td>
<td>554</td>
</tr>
<tr>
<td>Rotate 3D Objects</td>
<td>556</td>
</tr>
<tr>
<td>Scale 3D Objects</td>
<td>558</td>
</tr>
<tr>
<td>Use Grips to Modify Solids and Surfaces</td>
<td>561</td>
</tr>
<tr>
<td>Use 3D Subobject Grips</td>
<td>561</td>
</tr>
<tr>
<td>Cycle Through and Filter Subobjects</td>
<td>564</td>
</tr>
<tr>
<td>Use Grips to Edit 3D Solids and Surfaces</td>
<td>567</td>
</tr>
<tr>
<td>Modify 3D Subobjects</td>
<td>570</td>
</tr>
<tr>
<td>Move, Rotate, and Scale 3D Subobjects</td>
<td>570</td>
</tr>
<tr>
<td>Modify Faces on 3D Objects</td>
<td>573</td>
</tr>
<tr>
<td>Modify Edges on 3D Objects</td>
<td>575</td>
</tr>
<tr>
<td>Modify Vertices on 3D Objects</td>
<td>579</td>
</tr>
<tr>
<td>Work with Complex 3D Solids and Surfaces</td>
<td>581</td>
</tr>
<tr>
<td>Display Original Forms of Composite Solids</td>
<td>582</td>
</tr>
<tr>
<td>Modify Composite Solids and Surfaces</td>
<td>583</td>
</tr>
<tr>
<td>Shell and Remove Redundancies in 3D Objects</td>
<td>585</td>
</tr>
<tr>
<td>Press or Pull Bounded Areas</td>
<td>587</td>
</tr>
<tr>
<td>Add Edges and Faces to Solids</td>
<td>588</td>
</tr>
<tr>
<td>Modify Properties of 3D Solid, Surface, and Mesh</td>
<td>590</td>
</tr>
<tr>
<td>Modify Surfaces</td>
<td>594</td>
</tr>
<tr>
<td>Overview of Modifying Surfaces</td>
<td>594</td>
</tr>
<tr>
<td>Trim and Untrim Surfaces</td>
<td>596</td>
</tr>
<tr>
<td>Extend a Surface</td>
<td>598</td>
</tr>
<tr>
<td>Fillet a Surface</td>
<td>599</td>
</tr>
<tr>
<td>Edit NURBS Surfaces</td>
<td>600</td>
</tr>
<tr>
<td>Analyze Surfaces</td>
<td>602</td>
</tr>
<tr>
<td>Modify Mesh Objects</td>
<td>610</td>
</tr>
<tr>
<td>Overview of Modifying Meshes</td>
<td>610</td>
</tr>
<tr>
<td>Change Mesh Smoothness Levels</td>
<td>615</td>
</tr>
<tr>
<td>Refine Mesh Objects or Subobjects</td>
<td>619</td>
</tr>
<tr>
<td>Add Creases to Mesh</td>
<td>621</td>
</tr>
<tr>
<td>Modify Mesh Faces</td>
<td>622</td>
</tr>
<tr>
<td>Create and Close Mesh Gaps</td>
<td>626</td>
</tr>
<tr>
<td>Tips for Working with Mesh</td>
<td>627</td>
</tr>
</tbody>
</table>

Chapter 24 Create Sections and 2D Drawings from 3D Models | 635
| Work with Sections | 635 |
| Overview of Section Objects | 635 |
| Create Section Objects | 637 |
| Modify a Section View | 640 |
| Save and Publish Section Objects | 647 |
| Create a Flattened View | 650 |

Annotate Drawings | 653

Chapter 25 Work with Annotations | 655
<p>| Overview of Annotations | 655 |</p>
<table>
<thead>
<tr>
<th>Contents</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Scale Annotations</td>
<td>656</td>
</tr>
<tr>
<td>Overview of Scaling Annotations</td>
<td>657</td>
</tr>
<tr>
<td>Set Annotation Scale</td>
<td>658</td>
</tr>
<tr>
<td>Create Annotative Objects</td>
<td>660</td>
</tr>
<tr>
<td>Display Annotative Objects</td>
<td>670</td>
</tr>
<tr>
<td>Add and Modify Scale Representations</td>
<td>671</td>
</tr>
<tr>
<td>Set Orientation for Annotations</td>
<td>673</td>
</tr>
<tr>
<td>Chapter 26 Hatches, Fills, and Wipeouts</td>
<td>675</td>
</tr>
<tr>
<td>Overview of Hatch Patterns and Fills</td>
<td>675</td>
</tr>
<tr>
<td>Specify Hatch and Fill Areas</td>
<td>680</td>
</tr>
<tr>
<td>Control the Appearance of Hatches</td>
<td>684</td>
</tr>
<tr>
<td>Choose a Hatch Pattern or Fill</td>
<td>685</td>
</tr>
<tr>
<td>Control the Hatch Origin Point</td>
<td>689</td>
</tr>
<tr>
<td>Control the Scale of Hatch Patterns</td>
<td>691</td>
</tr>
<tr>
<td>Set Property Overrides for Hatches and Fills</td>
<td>692</td>
</tr>
<tr>
<td>Control the Display of Hatch Boundaries</td>
<td>694</td>
</tr>
<tr>
<td>Control the Draw Order of Hatches and Fills</td>
<td>696</td>
</tr>
<tr>
<td>Modify Hatches and Fills</td>
<td>697</td>
</tr>
<tr>
<td>Modify Hatch Properties</td>
<td>697</td>
</tr>
<tr>
<td>Modify Hatch Alignment, Scale, and Rotation</td>
<td>698</td>
</tr>
<tr>
<td>Reshape a Hatch or Fill</td>
<td>699</td>
</tr>
<tr>
<td>Re-create the Boundary of a Hatch or Fill</td>
<td>701</td>
</tr>
<tr>
<td>Create a Blank Area to Cover Objects</td>
<td>702</td>
</tr>
<tr>
<td>Chapter 27 Notes and Labels</td>
<td>705</td>
</tr>
<tr>
<td>Overview of Notes and Labels</td>
<td>705</td>
</tr>
<tr>
<td>Create Text</td>
<td>706</td>
</tr>
<tr>
<td>Overview of Creating Text</td>
<td>706</td>
</tr>
<tr>
<td>Create Single-Line Text</td>
<td>708</td>
</tr>
<tr>
<td>Create Multiline Text</td>
<td>711</td>
</tr>
<tr>
<td>Create and Edit Columns in Multiline Text</td>
<td>722</td>
</tr>
<tr>
<td>Import Text from External Files</td>
<td>723</td>
</tr>
<tr>
<td>Create Leaders</td>
<td>723</td>
</tr>
<tr>
<td>Overview of Leader Objects</td>
<td>724</td>
</tr>
<tr>
<td>Create and Modify Leaders</td>
<td>725</td>
</tr>
<tr>
<td>Work with Leader Styles</td>
<td>728</td>
</tr>
<tr>
<td>Add Content to a Leader</td>
<td>729</td>
</tr>
<tr>
<td>Use Fields in Text</td>
<td>732</td>
</tr>
<tr>
<td>Insert Fields</td>
<td>733</td>
</tr>
<tr>
<td>Update Fields</td>
<td>735</td>
</tr>
<tr>
<td>Use Hyperlinks in Fields</td>
<td>736</td>
</tr>
<tr>
<td>Work with Text Styles</td>
<td>738</td>
</tr>
<tr>
<td>Overview of Text Styles</td>
<td>738</td>
</tr>
<tr>
<td>Assign Text Fonts</td>
<td>740</td>
</tr>
</tbody>
</table>
Collaborate with Others ........................................... 941

Chapter 34 Use the Internet for Collaboration .................. 943
Get Started with Internet Access ................................. 943
Work with Drawing Files over the Internet ................... 944
Open and Save Drawing Files from the Internet ............... 944
Share Drawing Files Internationally ............................. 945
Access Buzzsaw for Project Collaboration ...................... 947
Work with Xrefs over the Internet ............................... 948

Render Drawings .................................................. 949

Chapter 35 Draw 2D Isometric Views ......................... 951
Set Isometric Grid and Snap ..................................... 951
Draw Isometric Circles .......................................... 953

Chapter 36 Add Lighting to Your Model ....................... 955
Overview of Lighting ............................................. 955
Standard and Photometric Lighting Workflow ................ 958
Illuminate a Scene ................................................ 960
Guidelines for Lighting .......................................... 960
Use Point Lights .................................................. 962
Use Spotlights .................................................... 965
Use Weblights .................................................... 967
Use Distant Lights ................................................ 976
Assigning a Shape to a Light ................................... 977
Adjust and Manipulate Lights .................................... 978
Control the Display of Lights ................................... 978
Adjust Light Placement ......................................... 979
Control Light Properties ....................................... 982
Incorporate Luminaire Objects .................................. 987

Chapter 37 Materials and Textures .............................. 989
Overview of Materials .......................................... 989
Browse Material Library ....................................... 990

Chapter 38 Render 3D Objects for Realism .................... 993
Overview of Rendering .......................................... 993
Prepare a Model for Rendering ................................ 994
Understand Face Normals and Hidden Surfaces ............. 994
Minimize Intersecting and Coplanar Faces .................... 996
Get Information
Find the Information You Need

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

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

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

Access and Search the Product Help

The Help system in AutoCAD for Mac uses a Web browser and is available online and offline.

You can access the Help system by doing one of the following:

- Press Fn-F1 or Cmd-/.
  If you press Fn-F1 or Cmd-/ when a command is active, the appropriate help topic is opened in the Web browser. Otherwise, the landing page of the Help system is displayed.

- On the Mac OS menu bar, click Help ➤ AutoCAD for Mac Help.
  The landing page of the Help system is displayed.

- In a dialog box, help the Help or ‘?’ button.
  The help topic related to the dialog box is opened in the Web browser.
Navigate Help

Each page of the help system is divided into four main areas:

- **Header** - Contains the navigation links to the Home page along with links that represent the path to the current topic. Along with navigation links, the Search text box is also located in the header.

- **Left Side** - Along the left side of a page is the table of contents that allow you to navigate in the current guide. You can also find links sections on the current page as well as related topics in the documentation set. When on the Home page, the left side contains a listing of the guides in the current documentation set.

- **Middle** - The middle of the page contains the content for the current topic. When on the Home page, the middle of the page contains links to the main topics in the selected guide in the documentation set from the left side.

- **Right Side** - The right side of the page contains links that are related to the current topic. These links come from Autodesk.com and are available only when using the online version of the Help system.

Search Help

In the upper-right corner of each page is a Search text box. Enter a text string to search on, and click the Search button or press Enter to begin the search. The results of the search are displayed on a new page. The left side of the results page lists the books that a search result was found in, while the right side displays the results for the selected book. Click a book from the left side to see additional search results, or click a link from the search results to open the associated topic.

Learn the Product

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

More than 1,200 ATC sites are available worldwide to meet your needs for discipline-specific, locally based training.
Autodesk Official Training Courseware (AOTC) is technical training material developed by Autodesk. You can purchase AOTC from your local reseller or distributor, or you can order it online from the Autodesk Store.

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

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

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

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

View the Product Readme

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

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

- View the Readme
Join the Customer Involvement Program

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

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

See the following links for more information.

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

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

To turn the CIP on or off

1. On the menu bar, click Help ➤ Customer Involvement Program.
2. In the Customer Involvement Program dialog box, choose whether you want to start or stop participating.
3. Click OK.
Get Information from Drawings

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

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

Obtain General Drawing Information

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

This information includes the following:

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

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

See also:

- Enter System Variables on the Command Line on page 18
- Add Identifying Information to Drawings on page 59
- Extract Geometric Information from Objects on page 252
Quick Reference

Commands

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

SETVAR
Lists or changes the values of system variables.

STATUS
Displays drawing statistics, modes, and extents.

TIME
Displays the date and time statistics of a drawing.

System Variables

CDATE
Stores the current date and time in decimal format.

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

SAVENAME
Displays the file name and directory path of the most recently saved drawing.
The User Interface
Start a Command

Use the menu bar, Tool Sets palette, and Command Line to access many frequently used commands.

Parts of the User Interface

The default user interface displays palettes and bars around the drawing area. Also, several controls are displayed within the drawing area.

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

- Cmd-1 turns the Tool Sets palette on and off
- Cmd-2 turns the Content Libraries palette on and off
Cmd-3 turns the Command Line on and off
Cmd-4 turns the Layers palette on and off
Cmd-5 turns the Properties Inspector on and off
Cmd-6 turns the Status bar on and off
Cmd-7 turns the References Manager palette on and off
Cmd-8 turns the Materials Browser palette on and off
Cmd-9 is not assigned
Cmd-0 turns all palettes and bars on and off

You can dock palettes by dragging them to the edge of your screen until a blue line appears, and then dropping them into place. You can also undock them by dragging and dropping.

Quick Reference

Commands

CUI
Manages the customized user interface elements in the product.

OPTIONS
Customizes the program settings.

The Menu Bar

The menu bar contains common commands organized into logical categories.
Use the menu bar when learning the product, or browsing for a command.
Many, but not all commands are accessible from the menu bar. Less commonly used commands can be entered in the Command Line. All available commands are listed in the Help system under Command Reference.

Quick Reference

Commands

CUI

Manages the customized user interface elements in the product.

The Tool Sets Palette

The Tool Sets palette provides efficient access to AutoCAD commands.

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

- Tool flyouts
- Tool groups
- Tool sets
Tool Flyouts

Some of the tools on the Tool Sets palette have a flyout indicator.

Click and hold the flyout to display several options for that command.

Tool Groups

The tools on the Tool Sets palette are organized into tool groups. Click the arrow to display the entire tool group, which includes additional commands. To make the tool group stay visible, click the lock icon at the bottom of the tool group.

If you right-click the Tool Sets palette, a menu displays that you can use to turn off any tool groups that you don’t need.
Tool Sets

Click the Tool Sets button to display a list of alternate sets of commands based on your current tasks. For example, clicking the Annotation tool set replaces the commands in the Tool Sets palette with commands associated with dimensioning.

Cmd-1 turns the Tool Sets palette on and off.

TIP Use the CUI editor to customize any tool set, or create your own tool sets.

Quick Reference

Commands

CUI

Manages the customized user interface elements in the product.

The Command Line

The Command Line provides a fast way to enter commands and system variables directly using the keyboard.

Overview of Using the Command Line

By default, the Command Line is displayed in the lower-left corner of screen.
Using the keyboard, you can enter the following in the Command Line:

- A command or command abbreviation called a *command alias*
- The capitalized letters of an option for a command
- A setting called a *system variable* that controls how the program operates by default

Many advanced users prefer this method for speed. Also, the Command Line displays prompts and error messages.

Cmd-3 turns the Command Line on and off.

**Quick Reference**

**Commands**

- **COMMANDLINE**
  - Displays the Command Line window.
- **COMMANDLINEHIDE**
  - Hides the Command Line window.
- **CUI**
  - Manages the customized user interface elements in the product.
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 or its command alias in the input area of the Command Line, and then press Enter or Spacebar. The Command Line includes several controls.

For example, instead of entering `circle` to start the CIRCLE command, you can enter `c`. Command aliases 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 or system variable, type one or more of its beginning letters in the Command Line and press Tab to cycle through all the possibilities. Then, press Enter or Spacebar.

**NOTE** When Dynamic Input is turned on and is set to display dynamic prompts, you can enter commands and options in tooltips near the cursor. Dynamic Input can be turned on or off from the status bar.

Specify Command Options

When you enter a command in the Command Line, you see either a set of options, a dialog box, or a palette. To specify an option displayed in the Command line, enter the capitalized letters for the option. For example, when you enter `circle`, the following prompt is displayed:

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

You can specify the center point for the circle either by entering $X,Y$ coordinate values, or by using the pointing device to click a point in the drawing area.

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

You can repeat the previous command by pressing Enter or Spacebar.

To repeat a recently used command, right-click in the Command Line or click the drop-down arrow to the left of the command input area. This action displays a shortcut menu with a list of recently used commands.

You can also repeat a recently used command by cycling through the commands with Up Arrow and Down Arrow keys, and then pressing Enter.

To cancel any command in progress, press ESC.

See also:

- Use Dynamic Input on page 223
- Keyboard Shortcuts
- Create Command Aliases

Quick Reference

Commands

COMMANDLINE
  Displays the Command Line window.
COMMANDLINEHIDE
  Hides the Command Line window.
CUI
  Manages the customized user interface elements in the product.

Enter System Variables on the Command Line

System variables are settings that control how certain commands work.

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.

With system variables, you can

- Turn on or turn off features. For example, the GRIDMODE system variable turns the grid display on and off when you change the value.
Control the operation of a command. For example, the HPASSOC system variable controls whether hatch patterns are associative by default.

Retrieve stored information about the current drawing and about the program configuration. For example, CDATE is a read-only system variable that stores the current date in decimal format. You can display the values of read-only system variables, but you cannot change them.

Usually system variables are accessible from dialog boxes. You can change their values either in a dialog box, directly in the Command Line, or automatically in a script or custom program.

To change the setting of a system variable

1 In the Command Line, enter the system variable name. For example, enter pickadd to change the style for selecting objects, whether selecting objects automatically replaces the current selection set, or whether they are added to the current selection set.

2 If necessary, press Fn+F1 to view the documentation for that system variable.

3 Enter the setting that you want to use. In the example of PICKADD, enter 0, 1, or 2 to determine how you select multiple objects.

Quick Reference

Commands

COMMANDLINE
Displays the Command Line window.

COMMANDLINEHIDE
Hides the Command Line window.

CUI
Manages the customized user interface elements in the product.

SETVAR
Lists or changes the values of system variables.
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 in the Command Line and in a dialog box. In many cases, you can enter a hyphen before a command to suppress the dialog box and display prompts in 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 a dialog box is useful for familiar operation with earlier versions of the program, and for using script files. There may be slight differences between the options in the dialog box and those available in the Command Line.

These system variables also affect the display of dialog boxes:

- **ATTDIA** controls whether the INSERT command uses a dialog box for entering block attribute values.
- **EXPERT** controls whether certain warning dialog boxes are displayed.
- **FILEDIA** controls the display of dialog boxes used with commands that read and write files. For example, if FILEDIA is set to 1, SAVEAS displays the Save Drawing As dialog box. If FILEDIA is set to 0, SAVEAS displays prompts on the command line. The procedures in this documentation assume that FILEDIA is set to 1. Even when FILEDIA is set to 0, you can display a file dialog box by entering a tilde (~) at the first prompt.

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

Quick Reference

**Commands**

**COMMANDLINE**

Displays the Command Line window.

**COMMANDLINEHIDE**

Hides the Command Line window.
System Variables

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

EXPERT
Controls whether certain prompts are issued.

FILEDIA
Suppresses display of file navigation dialog boxes.

View and Edit Within the Command History

You can copy text from the Command History to repeat commands.

You can expand and collapse the Command History in the Command Line using the indicated control.

Within the Command History, use the Up Arrow and Down Arrow keys, the scroll bar, or other scrolling method to locate and then highlight previously entered commands, system variables, and text.

By default, pressing Cmd-C copies highlighted text to the Clipboard. Pressing Cmd-V pastes text from the Clipboard to the Command Line.

To copy all the text in the Command History to the Clipboard, right-click and select Copy History from the shortcut menu, or enter the COPYHIST command. To save commands automatically to a log file starting with the next command, enter the LOGFILEON command.

Quick Reference

Commands
COMMANDLINE
Displays the Command Line window.
COMMANDLINEHIDE
Hides the Command Line window.

COPYHIST
Hides the Command Line window.

LOGFILEOFF
Closes the command history log file opened by LOGFILEON.

LOGFILEON
Writes the contents of the command history to a file.

System Variables
LOGFILEMODE
Specifies whether the contents of the command history are written to a log file.

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

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

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

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

- Display the controls for a user-interface element such as a palette, the status bar, or the ViewCube
- Control the command in progress, including command options, object snaps, and canceling.
- Display a list of recent input or repeat the last command entered
- Cut, copy, and paste from the Clipboard
- Display a dialog box, such as Drafting Settings or Preferences
- Undo the last command entered

In the Application Preferences dialog box (the OPTIONS command), 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.

Quick Reference

Commands

CUI
  Manages the customized user interface elements in the product.

OPTIONS
  Customizes the program settings.
Control the Drawing Area Interface

The drawing area includes several tools and controls for viewing and drawing operations. You can adjust the display of these interface elements.

Interface Themes and Background Color

Many options are provided for customizing the look and feel of the product, including the color of the icons and the background color of the drawing area.

The default color of the icons and palettes are dark gray. If you prefer, you can change this theme to a light color.

The default background color of the drawing area is a medium gray, which is optimum for displaying objects with different colors. Nevertheless, some people prefer a white or a black background color depending on their tasks and preferences.

See also:

- Set Up the Drawing Area on page 43

To change the color of the user interface between dark and light

1. On the menu bar, click AutoCAD, and then Preferences.
2. In the Application Preferences dialog box, left column, click Look & Feel.
3. Under Interface Theme, click in the Themes box, and click either Dark or Light.
4. Click OK.
To change the background color of the drawing area in Model space

1. On the menu bar, click AutoCAD, and then Preferences.
2. In the Application Preferences dialog box, left column, click Look & Feel.
3. Under Interface Theme, click in the Model box, and then click a color, or click Select Color.
   The default dark gray background color has an RGB value of 33,40,48.
4. If you clicked Select Color, the Color Palette dialog box is displayed. At the top of the dialog box, click either Index Color, True Color, or Color Books, and then make your color selection. Click OK to exit the Select Color dialog box.
5. Click OK.

Quick Reference

Commands
OPTIONS
Customizes the program settings.

Cursors in the Drawing Area

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

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

In the following illustrations, these cursors are displayed in order.
You can change the size of the crosshairs and pickbox cursors in the Application Preferences dialog box by clicking Cursor & Selection (the OPTIONS command).

**Quick Reference**

**Commands**

OPTIONS

Customizes the program settings.

**System Variables**

CURSORSIZE

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

PICKBOX

Sets the object selection target height, in pixels.

**Selection Style**

Selecting objects conforms to a selection style that is common to most Mac applications.

Use click and drag to specify a rectangular selection area. Drag to the left for a crossing selection, or drag to the right for a window selection.

Each time you select one or more objects, it automatically clears the previous selection. To add objects to the previous selection, press SHIFT as you select them.

You can change the behavior of object selection in the Application Preferences dialog box by clicking Cursor & Selection (the OPTIONS command).

See also:

- Select Multiple Objects on page 295
Quick Reference

Commands
OPTIONS
  Customizes the program settings.

System Variables
PICKADD
  Controls whether subsequent selections replace the current selection set or add to it.
PICKDRAG
  Controls the method of drawing a selection window.

The UCS Icon

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

You can move or rotate the UCS with the UCS command. The UCS is useful in 2D, and essential in 3D because it controls features that include

- The angular orientation that defines horizontal and vertical
- The alignment and angle of the grid, and hatch patterns
- The origin and orientation for 2D and 3D coordinate entry
- The orientation of construction places, projection planes, and the Z-axis direction for many 3D operations
You can change the appearance of the UCS icon with the UCSICON command, Properties option. With this command, you can also control whether the UCS icon is visible.

See also:

- Understand the User Coordinate System (UCS) on page 210
- Specify Workplanes in 3D (UCS) on page 214
- Control the Display of the User Coordinate System Icon on page 221

Quick Reference

Commands

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

UCS
Manages user coordinate systems.

UCSICON
Controls the visibility and placement of the UCS icon.

Viewport Label Menus

Viewport label menus are located at the top-left corner of each viewport, and provide a convenient way of changing views and visual styles.
By default, text is displayed that shows the current viewport settings. For example, the text might be

[+][Top][2D Wireframe]

You can click within each of the three bracketed areas.

- Click + to display more options
- Click Top to choose between several standard and custom views
- Click 2D Wireframe to choose one of several visual styles. Most of the other visual styles are used for 3D visualization

See also:
- Save and Restore Views on page 84
- Use a Visual Style to Display Your Model on page 92

Quick Reference

Commands

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

VSCURRENT
Sets the visual style in the current viewport.

System Variables

VPCONTROL
Controls whether the Viewport label menus are displayed in all viewports.

The ViewCube Tool

The ViewCube tool is a handy tool to control the orientation of 3D views. This tool is available in most Autodesk products, and provides a common experience when you switch between products.
Alternatively, you can use the 3DORBIT command to drag 3D views, and right-click for additional 3D viewing options.

See also:
- Use 3D Navigation Tools on page 108

Quick Reference

Commands

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

The Coordinates Display

The coordinates display is located in the lower-right corner of the active viewport and displays the current location of the crosshair cursor in the drawing area.

The display of the coordinates in the active viewport can be toggled in the Units & Guides tab (Application Preferences dialog box).
Along with the coordinates displayed in the active viewport, you can also get the current location of the crosshair cursor in a tooltip near the cursor when dynamic input is turned on. For more information about dynamic input, see Use Dynamic Input on page 223.

See also:
- Use Dynamic Input on page 223
- Overview of Coordinate Entry on page 199

Quick Reference

Commands

OPTIONS
Customizes the program settings.

System Variables

VPCOORDDISPLAY
Controls whether the current coordinates value of the crosshair cursor are displayed in the lower-right corner of the active viewport.

Model Space and Layouts

There are two working environments, or spaces, in which you can work, model space and paper space layouts.

- Model space is used to create 2D drawings and 3D models
- Paper space is used to create layouts for plotting

While you can plot from model space, layouts are more convenient for scaling views, changing the location of views, and controlling the area and settings used in plotting.

To switch between model space and a layout, click the drop-down near the left side of the status bar.
See also:

- Quick Start for Layouts on page 137

**Quick Reference**

**Commands**

**MSPACE**

In a layout, switches from paper space to model space in a layout viewport.

**MVIEW**

Creates and controls layout viewports.

**PSPACE**

In a layout, switches from model space in a viewport to paper space.
Control Status, Layers, Properties, and Content

Use the Status bar, Layers palette, Properties Inspector, and Content palette to change which drafting aids are enabled, modify the layers in the current drawing, the properties of the current drawing or selected objects, and insert blocks from custom content libraries.

The Status Bar

The Status bar includes buttons that turn on and off various features. For example, this is where you can conveniently turn on and off the grid display, grid snap, object snap, dynamic input, and so on. The status bar also includes controls to display lineweights and object transparency. Several controls relate to the annotation scaling feature.
One of the most important controls on the status bar, highlighted in the illustration, changes the drawing area between model space and paper space layouts.

Click the disclosure triangle at the far right end of the status bar to display the second row of controls, which include settings and operations for 3D.

Cmd-6 turns the status bar on and off.

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

1. Right-click any empty area of the status bar.
2. In the status bar menu, click Display, and then any flyout.
3. Click any button name in the flyout to change whether it is displayed or hidden.

**Quick Reference**

*System Variables*

**STATUSBAR**

**The Layers Palette**

The Layers palette provides two ways to display and manage layers and layer properties.

The disclosure triangle in the Layers palette expands and compresses the Layers palette to display either

- All layers and layer properties in a matrix of information, or
- The current layer only
Display All Layers and Layer Properties

The layers and layer properties in a drawing can be displayed as a matrix, similar to a spreadsheet. Each row contains a layer and each column represents a layer property. You can click the Display Settings button at the bottom of the Layers palette to control which layer properties are displayed or hidden.

When undocked in this format, the Layers palette can display all layer information simultaneously at the cost of taking up space on the screen. Docking the Layers palette reduces the space it takes up, but you might have to scroll left and right to see all the properties.

Display the Current Layer Only

Under normal working conditions, the compressed format that displays only the name of the current layer is adequate and recommended.

Review Layer Properties

The Properties Inspector palette can be used to display all the properties of either the current layer, or a selected layer as a vertical list. To switch from displaying object properties, click the Layer Properties button at the top of the Properties Inspector palette.

Cmd-4 turns the Layers palette on and off.

See also:

- Work with Layers on page 165
To create a new layer
1 If necessary, click the disclosure triangle to expand the Layers palette.
2 At the bottom-left corner of the palette, click the + button.
3 Enter the name of the new layer in the highlighted text area, and then press Enter.

To change the current layer
1 On the Layers palette, click the layer drop-down.
2 Click the layer that you want to make the current layer.

There are several alternative methods. In the expanded Layers palette, you can right-click a layer to display a menu, or you can double-click on the layer name.

To filter the list of layers
1 If necessary, click the disclosure triangle to expand the Layers palette.
2 Enter one of more characters in the Search area at the bottom of the palette.
   Only the layers with the characters that you entered are displayed in the Layers palette. Wildcards are not available.
3 It is recommended that you delete the text in the Search area when you are done.
   You can click the magnifying glass icon to display and choose from a list of previous searches.

Quick Reference

Commands
LAYER
Manages layers and layer properties.

The Properties Inspector
With the Properties Inspector, you can display and change the settings and properties for objects and for layers.
You can perform the following actions:

- Specify the current default properties assigned to all new objects
- View and change the properties of one or more selected objects
- Specify the default properties of the current Layer

The key to controlling the information that appears in the Properties Inspector is choosing either the Object/Current button, or the Layer Properties button in the top-left corner of the palette.

For object properties, clicking either the Essentials button or the All button controls the number of properties displayed.

**Object/Current Properties Button**

The Properties Inspector with the Object/Current button clicked can complete one of three actions depending on what is selected.

- With no objects selected, it displays the default properties to be used for all new objects. You can change these defaults by clicking a property in the palette, and specifying a different value.
- With one object selected, it displays the properties for that object only, and you can change any of its properties.
- With more than one object selected, it can either display only the common properties shared by the objects, or all the properties. Any property that you change is applied to all the selected objects.
Layer Properties Button

The Properties Inspector provides an efficient way of displaying the properties and settings associated with the current layer, or a layer that you select in the Layers palette.

Cmd-5 turns the Properties Inspector on and off.

See also:
- Overview of Object Properties on page 161

Quick Reference

Commands

PROPERTIES
Controls properties of existing objects.

The Content Palette

The Content palette allows you to access and manage content libraries. From the Content palette you can
- Create custom content libraries to organize frequently used blocks
- Add and remove blocks from the Favorites library or a custom library
- Insert blocks from the current drawing, Favorites library, or a custom library

Content Libraries

Libraries are used to help organize and access the blocks that you frequently insert into a drawing. By default, there is no content available in the Content palette. Custom libraries are created and managed using the Manage Content Libraries dialog box. You add content to a library by referencing a saved DWG or DXF file, or the blocks contained in a saved DWG file.

After content has been added to a library, you can
- Insert a block into a drawing (see -INSERT)
- Add a block to the Favorites library
- Remove a block from a library
Search for a block in a library

**WARNING** If a drawing being referenced by Favorites or a custom library is moved, the reference is maintained but the associated block cannot be inserted into a drawing.

See also:
- Insert Blocks on page 407

**Quick Reference**

**Commands**

**CONTENT**
- Opens the Content palette.

**CONTENTCLOSE**
- Closes the Content palette.

**System Variables**

**CONTENTSTATE**
- Indicates whether the Content palette is open or closed.
Customize the Drawing Environment

You can change many window and drawing environment settings in the Application Preferences dialog box. For example, you can change how often a drawing is automatically saved to a temporary file, and you can link the program to folders containing files you use frequently.

**Set Interface Options**

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

**Set Up the Drawing Area**

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

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

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

Some settings affect how you work in the drawing area:

- **Color Scheme (Application Preferences dialog box, Look & Feel tab)**. You specify a dark or light color theme for the overall user interface. The settings affect the window frame background, status bar, title bar, and palettes.
- **Background Colors (Application Preferences dialog box, Look & Feel tab).**
  You specify the background colors used in the Model and named layouts.

- **UCS Icon and ViewCube (Application Preferences dialog box, Look & Feel tab).** You can specify the display options for the UCS icon and ViewCube in model space.

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

- **Clean Screen.** You can expand the drawing area to display only the menu bar with the Clean Screen button on the status bar. Press Ctrl-0 to restore the previous setup.

**Tooltips**

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

Tooltips are displayed for tools on the Tool Sets and other palettes in the user interface. Hover the cursor over the control to display the tooltip.

**See also:**

- User Interface Customization in the *Customization Guide*

**Quick Reference**

**Commands**

3DCONFIG (-3DCONFIG)

- Sets options that affect 3D display performance.

CLEANSCREENON

- Clears the screen of the menu bar and all palettes.

CLEANSCREENOFF

- Restores the state of the display before CLEANSCREENON was used.

OPTIONS

- Customizes the program settings.

VIEWRES

- Sets the resolution for objects in the current viewport.
**System Variables**

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

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

DRAGMODE
Controls the way dragged objects are displayed.

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

GRIPCOLOR
Controls the color of unselected grips.

GRIPHOT
Controls the color of selected grips.

GRIPS
Controls the display of grips on selected objects.

HELPPREFIX
Sets the file path for the Help system.

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

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

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

INTELLIGENTUPDATE
Controls the graphics refresh rate.

ISAVEBAK
Improves the speed of incremental saves, especially for large drawings.
ISAVEPERCENT
Determines the amount of wasted space tolerated in a drawing file.

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

LOCALE
Displays a code that indicates the current locale.

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

LOGFILEMODE
Specifies whether the contents of the command history are written to a log file.

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

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

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

PAPERUPDATE
Controls the display of a warning dialog box when attempting to print a layout with a paper size different from the paper size specified by the default for the plotter configuration file.

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

PICKAUTO
Controls automatic windowing at the Select Objects prompt.
PICKBOX
Sets the object selection target height, in pixels.

PICKDRAG
Controls the method of drawing a selection window.

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

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

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

QTEXTMODE
Controls how text is displayed.

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

SAVEFILE
Stores the current automatic save file name.

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

SAVETIME
Sets the automatic save interval, in minutes.

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

TDUSRTIMER
Stores the user-elapsed timer.
TOOLTIPMERGE
Combines drafting tooltips into a single tooltip.

VISRETAIN
Controls the properties of xref-dependent layers.

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

Specify the Behavior of Palettes
Palettes, such as Tool Set, status bar, and Reference Manager can be docked, displayed as icons, or floated.

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

- **Resize.** Drag an edge of the palette to change its size. If one or more palettes are docked, dragging one of the palettes adjusts the size of the other docked palettes.

- **Collapse to Icons.** You can collapse the display of all palettes, except command line and status bar, to a set of icons that are displayed along the left or right side of the screen. Click an icon to temporarily display the associated palette. (PALETTEICONON command)

- **Show as Palettes.** Expands all palettes that are currently collapsed as icons. The palettes are returned to their previous size and location. (PALETTEICONOFF command)

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

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

The placement of palettes can be changed by dragging them on screen. You can control the location a palette is docked by dragging it to the edge of the screen and dropping it when you see a blue bar. You can also drag and drop palettes on a palette that is already docked.

Use RESETBLOCK to return all palettes to their default placement.
Quick Reference

Commands

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

PALETTEICONON
Collapses all open palettes, except command line and status bar, to a small bar displayed along the left or right side of the screen.

PALETTEICONOFF
Restores the display of all palettes collapsed by PALETTEICONON.

RESETPALETTES
Resets all palettes to their default placement settings.

SHOWPALETTES
Restores the display of hidden palettes.

System Variables

PALETTEICONSTATE
Indicates whether palettes are in icon state.

SHOWPALETTESTATE
Indicates whether palettes were hidden by the HIDEPALETTES command or restored by the SHOWPALETTES command.

Customize Startup

Command line switches can be used to control how the program is started from the Terminal window or a shell script.

You can use command line switches to specify several options when you start the program. For example, you can run a script or start with a specified drawing template.
Command line switches are parameters you can use to create custom shell scripts to start AutoCAD for Mac in a specific. Valid switches are listed in the following table.

<table>
<thead>
<tr>
<th>Switch</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>-b</td>
<td>Designates a script to run after you start the program (b stands for batch process). Scripts can be used to set up drawing parameters in a new drawing file. An SCR file type is assumed.</td>
</tr>
<tr>
<td>-nologo</td>
<td>No AutoCAD for Mac logo screen</td>
</tr>
</tbody>
</table>

The syntax for using command line switches is

"pathname/AutoCAD" ["drawingname"] [-switch "name"]

When using a switch option, you must follow the switch with a space and then the name of a file, path, or view within quotation marks. For example, the following entry starts the program from a folder named AutoCAD 2011 for Mac with the drawing template arch1.dwt, restores a named view PLAN1, and executes a script file startup.scr.

“/Applications/Autodesk/AutoCAD 2011 for Mac/AutoCAD.app/Contents/MacOS/AutoCAD for Mac” -t “/templates/arch1” -b “startup”
Start and Save Drawings
All drawings start from either a default drawing template file or a custom drawing template file that you create. Drawing template files store default settings, styles, and additional data.

Overview of Starting a New Drawing

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

Choose Drawing Units

A drawing unit can equal one inch, one millimeter, or any other system of measurement. For more information about drawing units, see Determine the Units of Measurement on page 54.

Choose a Drawing Template File

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

Customize a Drawing Template File

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

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

Commands

NEW
  Creates a new drawing.

OPTIONS
  Customizes the program settings.

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

System Variables

MEASUREMENT
  Controls whether the current drawing uses imperial or metric hatch pattern and linetype files.

Specify Units and Unit Formats

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

Determine the Units of Measurement

Before you start to draw, you must decide what one drawing unit represents based on what you plan to draw. You can convert a drawing between systems of measurement by scaling it.

Every object you create is measured in drawing units. Before you start to draw, you must decide what one drawing unit will represent based on what you plan to draw. Then you create your drawing at actual size with that convention.

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

If you start a drawing in one system of measurement (imperial or metric) and then want to switch to the other system, use SCALE to scale the model geometry by the appropriate conversion factor to obtain correct distances and dimensions.

For example, to convert a drawing created in inches to centimeters, you scale the model geometry by a factor of 2.54. To convert from centimeters to inches, the scale factor is 1/2.54 or about 0.3937.

See also:
- Set the Scale for Dimensions on page 801

Quick Reference

Commands

UNITS
  Controls coordinate and angle display formats and precision.

SCALE
  Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling.

System Variables

LUNITS
  Sets linear units.

LUPREC
  Sets the display precision for linear units and coordinates.
MEASUREINIT

Controls whether a drawing you start from scratch uses imperial or metric default settings.

MEASUREMENT

Controls whether the current drawing uses imperial or metric hatch pattern and linetype files.

UNITMODE

Controls the display format for units.

**Set the Unit Format Conventions**

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

**Set Linear Units**

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

For example, here are three variations of dynamic input.

![Dynamic Input Variations](image)

**Set Angular Units**

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

If you use surveyor’s angles when specifying polar coordinates, indicate whether the surveyor’s angles are in the north, south, east, or west direction. For example, to enter the relative coordinates for a property line that is 72 feet,
8 inches long with a bearing of 45 degrees north, 20 minutes, 6 seconds east, enter @728'8"<n45d20'6"e.

**Understand Rounding and Precision**

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

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

**Quick Reference**

**Commands**

**UNITS**

Controls coordinate and angle display formats and precision.

**System Variables**

**ANGBASE**

Sets the base angle to 0 with respect to the current UCS.

**ANGDIR**

Sets the direction of positive angles.

**AUNITS**

Sets units for angles.

**AUPREC**

Sets the display precision for angular units and coordinates.

**LUNITS**

Sets linear units.

**LUPREC**

Sets the display precision for linear units and coordinates.
MEASUREINIT
Controls whether a drawing you start from scratch uses imperial or metric default settings.

MEASUREMENT
Controls whether the current drawing uses imperial or metric hatch pattern and linetype files.

UNITMODE
Controls the display format for units.

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

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

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

Create a Drawing Template File
When you need to create several drawings that use the same conventions and default settings, you can save time by creating or customizing a drawing template file instead of specifying the conventions and default settings each time you start. Conventions and settings commonly stored in template files include

- Unit format and precision on page 54
- Title blocks and borders on page 407
- Layer names on page 167
- Snap and Grid spacing on page 232
- Text styles on page 738
By default, drawing template files are stored in the template folder, where they are easily accessible. You can use the Application Preferences dialog box to set a default for both the template folder and the drawing template file.

**Quick Reference**

**Commands**

NEW
   Creates a new drawing.
OPEN
   Opens an existing drawing file.
OPTIONS
   Customizes the program settings.
SAVEAS
   Saves a copy of the current drawing under a new file name.

**System Variables**

MEASUREMENT
   Controls whether the current drawing uses imperial or metric hatch pattern and linetype files.

**Add Identifying Information to Drawings**

You can keep track of your drawings more easily if you add keywords or other information to them.
Use Finder
Finder can be used to location drawing files. For example, you can search for all files created on a certain date, or for files you modified yesterday.

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 732.

Quick Reference
System Variables
CDATE
Stores the current date and time in decimal format.
DATE
Stores the current date and time in Modified Julian Date format.
TDCREATE
Stores the local time and date the drawing was created.
TDINDWG
Stores the total editing time, which is the total elapsed time between saves of the current drawing.
TDUCREATE
Stores the universal time and date that the drawing was created.
TDUPDATE
Stores the local time and date of the last update/save.
TDUUPDATE
Stores the universal time and date of the last update or save.
Open or Save a Drawing

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

Open a Drawing

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

To open a drawing, you can

- Use Open on the File menu to display the Select File dialog box. If the FILEDIA system variable is set to 0, the Command prompt version displays instead of a file navigation dialog box.
- Double-click a drawing in Finder to launch AutoCAD for Mac® and open the drawing. If the program is already running, the drawing opens in the current session.
- Drag a drawing from Finder onto the AutoCAD for Mac icon in the Dock. 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.

Work on Drawings During Loading

You can work on drawings before they are fully open. This is useful when you work on large drawings and you want to begin working immediately. To take advantage of this capability, three conditions are required.

- The drawing must have been saved in paper space.
The INDEXCTL system variable must be set to a non-zero value.

When these conditions are met, you can create or modify visible objects, pan or zoom, turn off or freeze layers, and any other operation that does not require displaying objects not visible when the drawing was last saved.

NOTE The Quick View feature will not be fully functional during loading under these conditions.

Resolve Missing References

As you open a drawing, you are notified (messages and task dialog boxes) when a reference cannot be located. From the References - Unresolved Reference Files task dialog box, click Update the Location of the Referenced Files to open the Reference Manager palette to make changes to missing external references.

The following table outlines some of the references that might be missing and describes how to handle them.

<table>
<thead>
<tr>
<th>Missing Reference Types</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>External references</td>
<td>Missing external references are the result of AutoCAD for Mac not being able to resolve the last known location of an xref, raster image, or underlay. To resolve a missing external reference, locate the file and update its location using the Reference Manager palette. For information about resolving missing referenced files, see</td>
</tr>
<tr>
<td></td>
<td>■ Resolve Missing External References on page 898</td>
</tr>
<tr>
<td></td>
<td>■ Attach Raster Image Files on page 908</td>
</tr>
<tr>
<td>Shapes</td>
<td>Missing shape files are often the result of custom shapes being used in a linetype. Browse to the missing linetype file, or place the shape file in the folder with the drawing or one of the support paths defined in the Options dialog box. For information about custom shape files in linetypes, see Shapes in Custom Line-types in the Customization Guide.</td>
</tr>
</tbody>
</table>
Work with Large Objects
AutoCAD 2010 supports object size limits greater than those available in previous releases. With increased object size limits you can create larger and more complex models. Using increased object size limits can result in compatibility issues with legacy drawing file formats (AutoCAD 2007 and earlier).

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

Recover Defective Drawing Files
In some circumstances, it is possible that a drawing file becomes defective. This can result from hardware problems or transmission errors. If a drawing file is corrupt, you might be able to recover it. See Repair, Restore, or Recover Drawing Files on page 75.

Change the Default Drawing Folder
Each time you start AutoCAD for Mac, the Documents folder is the default path in each standard file selection dialog box.

Alternatively, you can start AutoCAD for Mac in the current folder from the Terminal window. Set REMEMBERFOLDERS to 0 and then start AutoCAD for Mac from the current folder.

See also:
- Open and Save Drawing Files from the Internet on page 944

Quick Reference
CLOSE
Closes the current drawing.
CLOSEALL
Closes all currently open drawings.
OPEN
Opens an existing drawing file.
OPTIONS
Customizes the program settings.

WHOHAS
Displays ownership information for opened drawing files.

DWGCHECK
Checks drawings for potential problems when opening them.

FILEDIA
Suppresses display of file navigation dialog boxes.

INDEXCTL
Controls whether layer and spatial indexes are created and saved in drawing files.

LARGEOBJECTSUPPORT
Controls large object size limit support when you open and save drawings.

REMEMBERFOLDERS
Controls the default path displayed in standard file selection dialog boxes.

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

Work with Multiple Open Drawings
You can preview and switch between open drawings and layouts in a drawing and transfer information between open drawings.

Preview Open Drawings and Layouts
With QuickView, you can easily preview and switch between open drawings and layouts in an open drawing.

The Show Drawings & Layouts button on the status bar allows you to do the following:

- **Open drawings.** All copen drawings are displayed along the left side of the QuickView dialog box. Double-click a drawing thumbnail to switch to the corresponding drawing file, or right-click a drawing thumbnail to display
a list of available options. For more information, see Switch Between Open Drawings on page 65.

- **Layouts in an open drawing.** Displays the Model layout and named layouts in the current drawing or the selected drawing when there is more than one drawing open. Double-click a layout thumbnail to switch to the corresponding layout and drawing file, or right-click a layout thumbnail to display a list of available options. For more information, see Switch Between Layouts in the Current Drawing on page 66.

**Quick Reference**

**QUICKVIEW**
Displays a list of all open drawings, and the layouts contained in the current drawing or the selected drawing when more than one drawing is open.

**STATUSBAR**
Controls the display of the status bar.

**Switch Between Open Drawings**
Switch between open drawings.

You can use one of the following methods to switch between open drawings:

- On the menu bar, click the Window menu and choose a drawing from the bottom of the menu.

- In the Mac OS Dock, right-click the AutoCAD for Mac icon and choose a drawing from the top of the menu.

- In the Mac OS Dock, right side, click the thumbnail that represents the open drawing.

- On the status bar, click the Show Drawings & Layouts button. In the QuickView dialog box, click the drawing thumbnail along the left side.

- On the status bar, click the Drawings & Layouts pop-up menu and choose a drawing from the top of the menu.
Quick Reference

QUICKVIEW
Displays a list of all open drawings, and the layouts contained in the current drawing or the selected drawing when more than one drawing is open.

UPDATETHUMBSNOW
Manually updates thumbnail previews for named views, drawings, and layouts.

UPDATETHUMBNAIL
Controls updating of the thumbnail previews for views and layouts.

Switch Between Layouts in the Current Drawing
Switch between the model space and layouts in the current drawing.
You can use one of the following methods to switch between layouts in the current drawing:

■ On the status bar, click the Show Drawings & Layouts button. In the QuickView dialog box, click the layout thumbnail on the right side.

■ On the status bar, click the Drawings & Layouts pop-up menu and choose a layout from the middle section of the menu.

■ At the Command prompt, enter `ctab` and press Enter. When prompted for a new value, enter `model` or the name of a layout in the drawing and press Enter.

Quick Reference

LAYOUT
Creates and modifies drawing layouts.

QUICKVIEW
Displays a list of all open drawings, and the layouts contained in the current drawing or the selected drawing when more than one drawing is open.
UPDATETHUMBSNOW
Manually updates thumbnail previews for named views, drawings, and layouts.

CTAB
Returns the name of the current layout in the drawing.

UPDATETHUMBNAI
Controls updating of the thumbnail previews for views and layouts.

Transfer Information between Open Drawings
You can easily transfer information between drawings that are open in a single session.

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

■ Reference other drawings.
■ Copy and paste between drawings.
■ Use object snaps and the Copy with Basepoint (COPYBASE) command to ensure accurate placement.

Quick Reference

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

Save a Drawing
You save drawing files for later use just as you do with other applications. You can also set up automatic saving and backup files and save only selected objects.

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

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

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

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

### Save Part of a Drawing File

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

### Save to a Different Type of Drawing File

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

### Save with Visual Fidelity for Annotative Objects

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

If you work primarily in model space, it is recommended that you turn off visual fidelity (set SAVEFIDELITY to 0). However, if you need to exchange drawings with other users, and layout fidelity is most important, then visual fidelity should be turned on (set SAVEFIDELITY to 1).

**NOTE** The SAVEFIDELITY system variable does not effect saving a drawing to the AutoCAD 2010 drawing or DXF file formats.

Annotative objects may have multiple scale representation. When visual fidelity is on, annotative objects are decomposed and scale representations are saved (in an *anonymous block*) to separate layers, which are named based on their original layer and appended with a number. If you explode the block in AutoCAD 2007 or earlier releases, and then open the drawing in AutoCAD 2008 or later releases, each scale representation becomes a separate annotative object, each with one annotation scale. It is not recommended that you edit or create objects on these layers when working with a drawing created in AutoCAD 2008 and later releases in AutoCAD 2007 and earlier releases.
When this option is not selected, a single model space representation is displayed on the Model layout. More annotation objects may be displayed on the Model layout depending on the ANNOALLVISIBLE setting. Also, more objects may be displayed in paper space viewports at different sizes than in AutoCAD 2008 and later releases.

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

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

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

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

**Work Internationally**

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

If a drawing file is created in a different language version of the operating system, the following will occur:

- If support for the language is installed, the file name characters are visible in Finder.
- If support for the language is not installed, the file name characters appear as a series of boxes in Finder.

In either case, you will be able to open the drawing file beginning with AutoCAD 2007 because the product is Unicode-compliant.

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

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

When saving to a legacy drawing file format (AutoCAD 2007 or earlier), the drawing cannot contain large objects; there might be compatibility issues with trying to open the drawing. The LARGEOBJECTSUPPORT system variable controls the large object size limits used and the warning messages displayed when a drawing is saved.

The following explains how object size limits for drawings is determined:

- Drawing files cannot exceed an internal size limit of 4GB. This size is based on the total size of all objects in a drawing when uncompressed. Since a drawing file is normally compressed, the final size of a saved drawing file on disk will vary based on the size and number of objects in a drawing.

- Each individual object in a drawing cannot exceed an uncompressed size limit of 256MB. For example, a mesh object, when saved to a file and compressed, might be 75MB in size while the same object when uncompressed might be 257MB.

In these situations, the drawing cannot be saved to an AutoCAD 2007 or earlier file format until the issues are resolved. You can resolve the size limits by breaking the drawing or objects up into several drawings or objects.

See also:

- Save Drawings to Previous Drawing File Formats on page 931
- Work with Drawings in Earlier Releases on page 928
- Export Drawings to Other File Formats on page 923
- Create Drawing Files for Use as Blocks on page 414
- Add Identifying Information to Drawings on page 59
- Create and Restore Backup Files on page 77
- Share Drawing Files Internationally on page 945
- Open and Save Drawing Files from the Internet on page 944
Quick Reference

**BLOCK**
Defines a block definition from selected objects.

**OPTIONS**
Customizes the program settings.

**QSAVE**
Saves the current drawing using the specified default file format.

**QUIT**
Exits the program.

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

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

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

**DWGCHECK**
Checks drawings for potential problems when opening them.

**DWGNAME**
Stores the name of the current drawing.

**DWGPREFIX**
Stores the drive and folder prefix for the drawing.

**DWGTITLED**
Indicates whether the current drawing has been named.

**FILEDIA**
Suppresses display of file navigation dialog boxes.

**ISAVEBAK**
Improves the speed of incremental saves, especially for large drawings.
ISAVEPERCENT
Determines the amount of wasted space tolerated in a drawing file.

LARGEOBJECTSUPPORT
Controls large object size limit support when you open and save drawings.

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

SAVEFIDELITY
Controls whether the drawing is saved with visual fidelity.

SAVEFILE
Stores the current automatic save file name.

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

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

SAVETIME
Sets the automatic save interval, in minutes.

Find a Drawing File
You can search for a drawing using name, location, and date filters.

■ Use Searchlight in Finder to search for drawings using name, location, and date filters.

■ Use the Select File dialog box for the OPEN command to display drawing file previews. When the RASTERPREVIEW system variable is on, a raster preview image is automatically generated and stored with the drawing when you save it.

See also:

■ Add Identifying Information to Drawings on page 59
Quick Reference

OPEN

Opens an existing drawing file.

Specify Search Paths and File Locations

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

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

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

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

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

If you want to use a file that contains custom interface elements, specify it in the Customizations Files item on the Application tab (Application Preferences dialog box).
Quick Reference

OPTIONS
  Customizes the program settings.

MYDOCUMENTSPREFIX
Repair, Restore, or Recover Drawing Files

If a drawing file is damaged or if your program terminates unexpectedly, you can recover some or all of the data by using commands to find and correct errors, or by reverting to a backup file.

Repair a Damaged Drawing File

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

Repair and Recovery

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

A drawing file is marked as damaged if corrupted data is detected, or if you request that the drawing be saved after a program failure. If the damage is minor, sometimes you can repair the drawing simply by opening it. A recovery notification is displayed while opening drawing files that are damaged and need recovery. You can

- **RECOVER.** Performs an audit on, and attempts to open, any drawing file.
- **AUDIT.** Finds and corrects errors in the current drawing.
- **RECOVERAUTO.** Controls the display of recovery notifications before or after opening a damaged drawing file.
Example: Auditing Files

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

Auditing Header
DXF NAME Current Value Validation Default
PDMODE 990 - 2040
UCSFOLLOW 811 or 0
Error found in auditing header variables
4 Blocks audited
Pass 1 4 objects audited
Pass 2 4 objects audited
Total errors found 2 fixed 2

If you chose not to correct the errors, the last statement changes to
Total errors found 2 fixed 0.

The output from a recovery audit is written to an audit log (ADT) file if the AUDITCTL system variable is set to 1 (On).

Recovery does not necessarily preserve the high-level consistency of the drawing file. The program extracts as much material as it can from the damaged file.

Quick Reference

Commands

AUDIT
Evaluates the integrity of a drawing and corrects some errors.

OPTIONS
Customizes the program settings.

RECOVER
Repairs and then opens a damaged drawing file.

System Variables

AUDITCTL
Controls whether AUDIT creates an audit report (ADT) file.
RECOVERAUTO
Controls the display of recovery notifications before or after opening a damaged drawing file.

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

Create and Restore Backup Files
Backup files help ensure the safety of your drawing data. If a problem occurs, you can restore a drawing backup file.

Computer hardware problems, power failures or surges, user mistakes, or software problems can cause errors in a drawing. By saving your work frequently, you can ensure a minimum of lost data if your system fails for any reason. If a problem occurs, you can restore a drawing backup file.

Use Backup Files
In the Application tab (Application Preferences dialog box), you can specify that backup files are created when you save drawings. If you do, each time you save a drawing, the previous version of your drawing is saved to a file with the same name and a .bak file extension. The backup file is located in the same folder as the drawing file.

You can revert to your backup version by renaming the .bak file in Finder 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

**Quick Reference**

**Commands**

OPTIONS
  Customizes the program settings.

**System Variables**

ISAVEBAK
  Improves the speed of incremental saves, especially for large drawings.
Control the Drawing Views
Change Views

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

Pan or Zoom a View

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

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

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

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

The options described here are the options most commonly used.
Zoom to Magnify a Specified Rectangular Area

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

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

Zoom in Real Time

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

Zoom to Magnify One or More Objects

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

Zoom to View All Objects in the Drawing

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

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

Commands

PAN
Moves the view planar to the screen.

UNDO
Reverses the effect of commands.

VIEWRES
Sets the resolution for objects in the current viewport.

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

System Variables

EXTMAX
Stores the upper-right point of the drawing extents.

See also:

■ Scale Views in Layout Viewports on page 147
EXTMIN
Stores the lower-left point of the drawing extents.

MBUTTONPAN
Controls the behavior of the third button or wheel on the pointing device.

RTDISPLAY
Controls the display of raster images and OLE objects during Realtime ZOOM or PAN.

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

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

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

Named views are saved with a drawing and can be used any time. When you are composing a layout, you can restore a named view to a viewport on the layout.

Save a View
When you name and save a view, the following settings are saved:

- Magnification, center point, and view direction
- View category that you assign to the view (optional)
- The location of the view (the Model or a specific named layout)
- Layer visibility in the drawing at the time the view is saved
- User coordinate system
- 3D perspective
- Live section
- Visual style

**Restore a Named View**
You can use named views to do the following:
- Restore a view that you use frequently while working in model space.
- Restore a view on a layout that is zoomed into an area of interest on the layout.
- With multiple model or layout viewports, restore a different view in each one.

**Quick Reference**

**Commands**
**VIEW**
- Saves and restores named model space views, layout views, and preset views.

**VPORTS**
- Creates multiple viewports in model space or paper space.

**ZOOM**
- Increases or decreases the magnification of the view in the current viewport.

**Control the 3D Projection Style**
You can view both parallel and perspective projection of a 3D model.

**Overview of Parallel and Perspective Views**
You can create realistic visual effects in a drawing by defining either parallel or perspective projections of a model.

The difference between perspective views and parallel projections is that perspective views require a distance between a theoretical camera and target point. Small distances produce severe perspective effects; large distances produce mild effects.
The following illustration shows the same model in both a parallel projection and perspective projection. Both are based on the same viewing direction.

Quick Reference

Commands

3DORBIT
Rotates the view in 3D space, but constrained to horizontal and vertical orbit only.

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

System Variables

BACKZ
Stores the back clipping plane offset from the target plane for the current viewport, in drawing units.

FRONTZ
Stores the front clipping plane offset from the target plane for the current viewport, in drawing units.

LENSLENGTH
Stores the length of the lens (in millimeters) used in perspective viewing.

TARGET
Stores the location (as a UCS coordinate) of the target point for the current viewport.

VIEWDIR
Stores the viewing direction in the current viewport, expressed in UCS coordinates.
VIEWMODE
Stores the View mode for the current viewport.

VIEWTWIST
Stores the view rotation angle for the current viewport measured relative to the WCS.

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

Define a Perspective Projection (DVIEW)
Perspective projections require a distance between a theoretical camera and a target point. Small distances produce severe perspective effects; large distances produce milder effects.

A perspective view remains in effect until the perspective effect is turned off or until a new view is defined in its place.

Quick Reference

Commands

3DORBIT
Rotates the view in 3D space, but constrained to horizontal and vertical orbit only.

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

System Variables

BACKZ
Stores the back clipping plane offset from the target plane for the current viewport, in drawing units.

FRONTZ
Stores the front clipping plane offset from the target plane for the current viewport, in drawing units.
LENSLENGTH
Stores the length of the lens (in millimeters) used in perspective viewing.

PERSPECTIVE
Specifies whether the current viewport displays a perspective view.

PERSPECTIVECLIP
Determines the location of eyepoint clipping.

TARGET
Stores the location (as a UCS coordinate) of the target point for the current viewport.

VIEWDIR
Stores the viewing direction in the current viewport, expressed in UCS coordinates.

VIEWMODE
 Stores the View mode for the current viewport.

VIEWTWIST
 Stores the view rotation angle for the current viewport measured relative to the WCS.

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

**Define a Parallel Projection**
You can define a parallel projection.

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

- Enter a coordinate or angles that represent your viewing location in 3D.
- Change to a view of the XY plane of the current UCS, a saved UCS, or the WCS.
- Change the 3D view dynamically with your pointing device.
- Set front and back clipping planes to limit the objects being displayed.
Viewing in 3D is available only in model space. If you are working in paper space, you cannot use 3D viewing commands such as VPOINT, DVIEW, or PLAN to define paper space views. The view in paper space is always a plan view.

**Quick Reference**

**Commands**

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

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

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

**System Variables**

**PERSPECTIVE**
- Specifies whether the current viewport displays a perspective view.

**VIEWDIR**
- Stores the viewing direction in the current viewport, expressed in UCS coordinates.

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

**Choose Preset 3D Views**

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

A quick way to set a view is to choose one of the predefined 3D views. You can select predefined standard orthographic and isometric views by name or description. These views represent commonly used options: Top, Bottom, Front, Left, Right, and Back. In addition, you can set views from isometric...
options: SW (southwest) Isometric, SE (southeast) Isometric, NE (northeast) Isometric, and NW (northwest) Isometric.

To understand how the isometric views work, imagine you are looking down at the top of a box. If you move toward the lower-left corner of the box, you are viewing the box from the SW Isometric View. If you move toward the upper-right corner of the box, you are viewing it from NE Isometric View.

Quick Reference

Commands

VIEW

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

Define a 3D View with Coordinate Values or Angles

You can define a viewing direction by entering the coordinate values of a point or the measures of two angles of rotation.

This point represents your position in 3D space as you view the model while looking toward the origin (0,0,0). Viewpoint coordinate values are relative to the world coordinate system unless you change the WORLDVIEW system variable. The conventions for defining standard views differ between architectural (AEC) and mechanical design. In AEC design, the perpendicular view of the XY plane is the top or plan view; in mechanical design, the perpendicular view of the XY plane is the front view.

You can rotate a view using DDVPOINT. The following illustration shows a view defined by two angles relative to the X axis and the XY plane of the WCS.
Quick Reference

Commands

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

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

System Variables

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

Change to a View of the XY Plane

You can change the current viewpoint to a plan view of the current UCS, a previously saved UCS, or the WCS.

A plan view is a view aimed toward the origin (0,0,0) from a point on the positive Z axis. This results in a view of the XY plane.

You can restore the view and coordinate system that is the default for most drawings by setting the UCS orientation to World and then setting the 3D view to Plan View.
Quick Reference

Commands

DV\[e\]V\[E\]E
Defines parallel projection or perspective views by using a camera and target.

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

System Variables

BACKZ
Stores the back clipping plane offset from the target plane for the current viewport, in drawing units.

FRONTZ
Stores the front clipping plane offset from the target plane for the current viewport, in drawing units.

VIEWDIR
Stores the viewing direction in the current viewport, expressed in UCS coordinates.

VIEWTWIST
Stores the view rotation angle for the current viewport measured relative to the WCS.

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

Shade a Model and Use Edge Effects
Hiding lines enhances the drawing and clarifies the design. The addition of shading produces a more realistic image of your model.

Use a Visual Style to Display Your Model
Visual styles control the display of edges and shading a viewport.
Control the effect of a visual style by changing its properties. When you apply a visual style or change its settings, the associated viewport is automatically updated to reflect those changes.

The Properties Inspector displays all visual styles available in the drawing under the Visual Styles section.

The following predefined visual styles are supplied with the product:

- **2D Wireframe.** Displays objects using lines and curves to represent the boundaries.
  
  **NOTE** Raster images, linetypes, and lineweights are visible.

- **Conceptual.** Displays objects using smooth shading and the Gooch face style. The Gooch face style transitions between cool and warm colors, rather than dark and light. The effect is less realistic, but it can make the details of the model easier to see.

- **Hidden.** Displays objects using wireframe representation and hides lines representing back faces.

- **Realistic.** Displays objects using smooth shading and materials.

- **Shaded.** Displays objects using smooth shading.

- **Shaded with Edges.** Displays objects using smooth shading and visible edges.

- **Shades of Gray.** Displays objects using smooth shading and monochromatic shades of gray.

- **Sketchy.** Displays objects with a hand-sketched effect by using the Line Extensions and Jitter edge modifiers.

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

- **X-ray.** Displays objects with partial transparency.
In shaded visual styles, faces are lit by two distant light sources that follow the viewpoint as you move around the model. This default lighting is designed to illuminate all faces in the model so that they are visually discernable. Default lighting is available only when other lights, including the sun, are off.

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

**Quick Reference**

**Commands**

**SHADEMODE**

Starts the VSCURRENT command.

**VSCURRENT**

Sets the visual style in the current viewport.

**VSAVE**

Saves a visual style.
System Variables

VSLIGHTINGQUALITY

Sets the lighting quality in the current viewport.

Customize a Visual Style

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

Shade and Color Faces

Shading and color effects control the display of faces in a model.

Face Styles

The face style defines the shading on a face. Realistic (below left) is meant to produce the effect of realism. Gooch (below right) can show details better by softening the contrast between lit areas and shadowed areas. Lit areas use warm tones and shadowed areas use cool tones.
The None face style produces no shading, and displays only edges. Customize edge settings to control whether facet edges or isolines are displayed.

**Lighting Quality**

Lighting quality determines the smoothness of shaded objects.

Faceted lighting computes a single color for each face. Individual faces appear flat. Smooth lighting smoothes the edges between polygon faces by computing the colors as a gradient between the faces’ vertices. This gives objects a smooth appearance.

For the Smoothest option, the Per-Pixel Lighting setting must be enabled under the Hardware acceleration option of -3DCONFIG. The colors are
computed for individual pixels, giving a smoother appearance. If not, the Smooth setting is used instead.

**Highlights**

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

**Opacity**

The opacity property controls the transparency of objects.
Face Color Modes

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

Quick Reference

Commands

VSCURRENT

Sets the visual style in the current viewport.
VSSAVE
Saves a visual style.

System Variables

VSFACECOLORMODE
Controls how the color of faces is calculated.

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

VSFACEOPACITY
Turns on and off a preset level of transparency for 3D objects.

VSFACESTYLE
Controls how faces are displayed in the current viewport.

VSLIGHTINGQUALITY
Sets the lighting quality in the current viewport.

VSMONOCOLOR
Sets the color for monochrome and tint display of faces in the visual style applied to the current viewport.

Display Backgrounds and Shadows

The visual style also controls the display of backgrounds and shadows in the viewport.

Backgrounds

You can use a color, a gradient fill, an image, or the sun & sky as a background in the viewport in any 3D visual style, even one that does not shade objects. When Background is set to On in the current visual style, the background is displayed.

NOTE AutoCAD 2011 for Mac does not support the ability to create a named view with a background or assign a background to the current view. If a background is assigned to the current view or a named view, it does display in the current viewport.
Shadows

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

NOTE To display mapped object shadows, hardware acceleration is required. When Enhanced 3D Performance is off, mapped object shadows cannot be displayed. (To access these settings, enter `-3dconfig` at the Command prompt. Use options acceleration and then Hardware to get to the Enhanced 3D performance option.)

Displaying shadows can slow performance. You can turn off shadows in the current visual style while you work and turn them back on when you need them.

In the Properties Inspector, you can set the Shadow Display property for an object: casts shadows, receives shadows, casts and receives shadows, or ignores shadows.

More options are available for shadows used in rendering.

See also:

- Overview of Lighting on page 955
Quick Reference

Commands

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

VSCURRENT
Sets the visual style in the current viewport.

VSAVE
Saves a visual style.

System Variables

CSHADOW
Sets the shadow display property for a 3D object.

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

SHADOWPLANELOCATION
Controls the location of an invisible ground plane used to display shadows.

VSBACKGROUNDS
Controls whether backgrounds are displayed in the visual style applied to the current viewport.

VSSHADOWS
Controls whether a visual style displays shadows.

Control the Display of Edges

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

In a shaded or wireframe model, the visual style sets the visibility and appearance of isolines, facet edges, silhouette edges, occluded edges, and intersection edges. Facet edges (the edges between planar faces representing a surface) are displayed only when the angle between the facets is smaller than the crease angle value you specify.
Edge modifiers such as line extension and jitter, produce the appearance of a model that is still in the conceptual phase. Jitter makes lines appear as though they were sketched with a pencil. Line extension produces another kind of hand-drawn effect.

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

**Control the display of occluded lines in 2D View**

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

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

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

**NOTE** You can only change occluded color when the occluded lines are partially or completely visible.
IMPORTANT  After you have changed the settings for occluded lines, use the HIDE command to regenerate the drawing and display the changes.

Quick Reference

Commands
VSCURRENT
Sets the visual style in the current viewport.
VSSAVE
Saves a visual style.

System Variables
FACETRES
Adjusts the smoothness of shaded and rendered objects and objects with hidden lines removed.
INTERSECTIONCOLOR
Controls the color of polylines at the intersection of 3D surfaces when the visual style is set to 2D Wireframe.
INTERSECTIONDISPLAY
Specifies the display of intersection polylines.
VSEDGECOLOR
Sets the color of edges in the visual style in the current viewport.
VSEDGEJITTER
Makes edges on 3D objects appear wavy, as though they were sketched with a pencil.
VSEDGELEX
Makes edges on 3D objects extend beyond their intersection for a hand-drawn effect.
VSEDGEOVERHANG
Makes edges on 3D objects extend beyond their intersection for a hand-drawn effect.
VSEDGES
Controls the types of edges that are displayed in the viewport.

VSEDGESMooth
Specifies the angle at which crease edges are displayed.

VSHALOGAP
Sets the halo gap in the visual style applied to the current viewport.

VSINTERSECTIONEDGES
Controls the display of intersection edges in the visual style applied to the current viewport.

VSINTERSECTIONCOLOR
Specifies the color of intersection polylines in the visual style applied to the current viewport.

VSINTERSECTIONLTYPE
Sets the linetype for intersection lines in the visual style applied to the current viewport.

VSOBSCUREDDEDGES
Controls whether obscured (hidden) edges are displayed.

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

VSOBSCUREDLTYPE
Specifies the linetype of obscured (hidden) lines in the visual style applied to the current viewport.

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

VSOCCLUDEDDEDGES
Controls whether occluded (hidden) edges are displayed.

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

Controls display of silhouette edges of solid objects in the visual style applied to the current viewport.

VSSILHWIDTH

Specifies the width in pixels of silhouette edges in the current viewport.

**Control Performance**

3D graphics display and memory allocation can slow performance on your system. Performance and memory tuning are different approaches to delivering the best performance possible.

**Performance Tuning**

Performance tuning examines your graphics card and 3D display driver and determines whether to use software or hardware implementation for features that support both.

Features that cannot work properly on your system are turned off. Some features may work but not be recommended for use with your graphics card or 3D graphics display driver. Enable these features at your own risk. For information on the options available, see -3DCONFIG.

**Quick Reference**

**Commands**

3DCONFIG

Sets options that affect 3D display performance.

GRAPHICSCONFIG

Sets options for 3D display performance.

**System Variables**

VSSHADOWS

Controls whether a visual style displays shadows.
Memory Tuning

Performance can also be improved by adding memory to your system. This is especially true when working on larger models.

The system requirements for AutoCAD require at least 2 GB of physical memory (RAM) for working in 2D. For creating and working with 3D models, at least 4 GB of RAM is required.

The size and complexity of a model often defines how efficiently an application runs. If you notice increased hard drive activity, it means that physical memory has been exceeded and data is being passed to a swap file (virtual memory).

A swap file is an area on the hard drive that Windows uses as if it were physical memory (RAM). The swap file size is basically a limit which restricts the total virtual size of the AutoCAD process. A good rule of thumb for configuring your swap file is three times the amount of physical memory on your system. This usually sets the limit high enough that AutoCAD doesn’t run out of swap space.
Use Viewing Tools

When working in 3D, you'll often want to display different views so that you can see and verify the 3D effects in your drawing.

Specify 3D Views

You can control the 3D navigation display, projection, and visualization tools.

Overview of 3D Views

You can create an interactive view of your drawing in the current viewport.

Using the 3D viewing and navigation tools, you can navigate through a drawing. You can orbit, zoom, and swivel around a 3D model.

Quick Reference

Commands

3DDISTANCE

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

3DFORBIT

Rotates the view in 3D space without constraining roll.

3DORBIT

Rotates the view in 3D space, but constrained to horizontal and vertical orbit only.
3DORBITCTR
Sets the center of rotation in 3D Orbit view.

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

3DSWIVEL
Changes the target of the view in the direction that you drag.

3DZOOM
Zooms in and out in a perspective view.

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

System Variables
CAMERADISPLAY
Turns the display of camera objects on or off.

Use 3D Navigation Tools
3D navigation tools allow you to view objects in a drawing from different angles, heights, and distances.

Use the following 3D tools to orbit, swivel, adjust distance, zoom, and pan in a 3D view.

- **3D Orbit.** Moves around a target. The target of the view stays stationary while the camera location, or point of view, moves. The center of the viewport, not the center of the objects you’re viewing, is the target point.

- **Constrained Orbit.** Constrains 3D Orbit along the XY plane or the Z axis. (3DORBIT)

- **Free Orbit.** Orbits in any direction without reference to the planes. The point of view is not constrained along the XY plane of the Z axis. (3DFORBIT)

- **Adjust Distance.** Changes the distance of objects as you move the cursor vertically. You can make objects appear larger or smaller, and you can adjust the distance. (3DDISTANCE)
- **Swivel.** Simulates panning with a camera in the direction that you drag. The target of the view changes. You can swivel the view along the XY plane or along the Z axis. (3DSWIVEL)

- **Zoom.** Simulates moving the camera closer to an object or farther away. Zooming in magnifies the image. (3DZOOM)

- **Pan.** Starts the interactive 3D view and enables you to drag the view horizontally and vertically. (3DPAN)

## Quick Reference

### Commands

- **3DDISTANCE**
  Starts the interactive 3D view and makes objects appear closer or farther away.

- **3DFORBIT**
  Rotates the view in 3D space without constraining roll.

- **3DORBIT**
  Rotates the view in 3D space, but constrained to horizontal and vertical orbit only.

- **3DORBITCTR**
  Sets the center of rotation in 3D Orbit view.

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

- **3DSWIVEL**
  Changes the target of the view in the direction that you drag.

- **3DZOOM**
  Zooms in and out in a perspective view.

### System Variables

- **PERSPECTIVE**
  Specifies whether the current viewport displays a perspective view.
PERSPECTIVECLIP

Determines the location of eyepoint clipping.

**Create a 3D Dynamic View (DVIEW)**

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

With dynamic viewing, you can display the effects of changing your viewpoint as you make the changes. Using this method, you can also simplify your view temporarily by choosing only the objects that you need to determine the view. Alternatively, if you press Enter without selecting any objects, 3D Dynamic View displays a model of a small house instead of your actual drawing. You can use this house to define the viewing angle and distance. When your adjustments are complete and you exit the command, the changes are applied to the entire 3D model in the current view.

**NOTE** More powerful options for dynamic viewing in 3D are available in the 3DORBIT command. For more information, see Use 3D Navigation Tools on page 108.

**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:

![Position of clipping planes](image)

**NOTE** You can also set clipping planes when you create a camera glyph. For more information, see Change Camera Properties.
Quick Reference

Commands

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

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

System Variables

VIEWDIR
Stores the viewing direction in the current viewport, expressed in UCS coordinates.

VIEWTWIST
Stores the view rotation angle for the current viewport measured relative to the WCS.

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

Use ViewCube

ViewCube provides visual feedback of the current orientation of a model. You can use the ViewCube tool to adjust the viewpoint of the model.

Overview of ViewCube

ViewCube tool is a navigation tool that is displayed when you are working in 2D model space or 3D visual style. With ViewCube tool, you can switch between standard and isometric views.

The ViewCube tool is a persistent, clickable and draggable interface that you use to switch between standard and isometric views of your model. When you display the ViewCube tool, it is shown in one of the corners of the window over the model in an inactive state. The ViewCube tool provides visual feedback about the current viewpoint of the model as view changes occur. When the cursor is positioned over the ViewCube tool, it becomes active. You can drag
or click the ViewCube, switch to one of the available preset views, roll the current view, or change to the Home view of the model.

Control the Appearance of ViewCube

The ViewCube tool is displayed in one of two states: inactive and active. When the ViewCube tool is inactive, it appears partially transparent by default so that it does not obscure the view of the model. When active, it is opaque and may obscure the view of the objects in the current view of the model.

In addition to controlling the inactive opacity level of the ViewCube tool, you can also control the following properties for the ViewCube tool:

- Size
- Position
- Display of the UCS menu
- Default orientation
- Compass display

Using the Compass

The compass is displayed below the ViewCube tool and indicates which direction North is defined for the model. You can click a cardinal direction letter on the compass to rotate the model, or you can click and drag one of
the cardinal direction letters or the compass ring to interactively rotate the model around the pivot point.

Quick Reference

Commands

NAVVCUBE

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

System Variables

NAVVCUBEDISPLAY

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

NAVVCUBELOCATION

Identifies the corner in a viewport where the ViewCube tool is displayed.

NAVVCUBEOPACITY

Controls the opacity of the ViewCube tool when inactive.

NAVVCUBESIZE

Specifies the size of the ViewCube tool.

ViewCube Menu

Use the ViewCube menu to restore and define the Home view of a model, switch between view projection modes, and change the interactive behavior and appearance of the ViewCube tool.
The ViewCube menu has the following options:

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

- **Parallel.** Switches the current view to parallel projection.

- **Perspective.** Switches the current view to perspective projection.

- **Perspective with Ortho Faces.** Switches the current view to perspective projection unless the current view aligns with a face view defined on the ViewCube tool.

- **Set Current View as Home.** Defines the Home view of the model based on the current view.

- **ViewCube Settings.** Displays the dialog box where you can adjust the appearance and behavior of the ViewCube tool.

- **Help.** Launches the online Help system and displays the topic for the ViewCube tool.

**To display the ViewCube menu**

To display the ViewCube menu, do one of the following:

- Right-click on the compass, Home icon, or the main area of the ViewCube tool.

- Click the context menu button located below the ViewCube tool.

**Quick Reference**

**Commands**

```
NAVVCUBE
```

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

**System Variables**

```
PERSPECTIVE
```

Specifies whether the current viewport displays a perspective view.
Reorient the View of a Model with ViewCube

The ViewCube tool offers many intuitive ways to reorient the view of a model.

Reorient the Current View

You can reorient the current view of a model by clicking predefined areas on the ViewCube tool or dragging the ViewCube tool.

The ViewCube tool provides twenty-six defined parts to click and change the current view of a model. The twenty-six defined parts are categorized into three groups: corner, edge, and face. Of the twenty-six defined parts, six of the parts represent standard orthogonal views of a model: top, bottom, front, back, left, and right. Orthogonal views are set by clicking one of the faces on the ViewCube tool.

You use the other twenty defined parts to access angled views of a model. Clicking one of the corners on the ViewCube tool reorients the current view of the model to a three-quarter view, based on a viewpoint defined by three sides of the model. Clicking one of the edges reorients the view of the model to a half view based on two sides of the model.

The outline of the ViewCube tool helps you identify the form of orientation it is in: standard or fixed. When the ViewCube tool is in standard orientation, not orientated to one of the twenty-six predefined parts, its outline is displayed as dashed. The ViewCube tool is outlined in a solid continuous line when it is constrained to one of the predefined views.

Drag or Click the ViewCube Tool

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

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

Roll a Face View

When you view a model from one of the face views, two roll arrow buttons are displayed near the ViewCube tool. Use the roll arrows to rotate the current view 90 degrees clockwise or counterclockwise around the center of the view.

Switch to an Adjacent Face

When the ViewCube tool is active while viewing a model from one of the face views, four orthogonal triangles are displayed near the ViewCube tool. You use these triangles to switch to one of the adjacent face views.
Quick Reference

Commands

NAVVCUBE

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

Set the View Projection Mode

View projection produces realistic visual effects of a model.

The ViewCube tool supports two view projection modes (Perspective and Orthographic) and a combination of both these modes (Perspective with Ortho faces). Orthographic projection is also referred to as parallel projection. Perspective projected views are calculated based on the distance from a theoretical camera and target point. The shorter the distance between the camera and the target point, the more distorted the perspective effect appears; greater distances produce less distorted affects on the model. Orthographic projected views display all the points of a model being projected parallel to the screen.

Orthographic projection mode makes it easier to work with a model due to all the edges of the model appearing as the same size, regardless of the distance from the camera. Orthographic projection mode though, is not how you commonly see objects in the real world. Objects in the real world are seen in perspective projection. So when you want to generate a rendering or hidden line view of a model, using perspective projection will give the model a more realistic look.

The following illustration shows the same model viewed from the same viewing direction, but with different view projections.

Parallel
Perspective
When you change the view for a model, the view is updated using the previous projection mode unless the current projection mode for the ViewCube tool is Perspective with Ortho Faces. The Perspective with Ortho Faces mode forces all views to be displayed in perspective projection unless the model is being viewed from one of the face views: top, bottom, front, back, left, or right.

**Quick Reference**

**Commands**

NAVVCUBE

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

**System Variables**

NAVVCUBEDISPLAY

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

PERSPECTIVE

Specifies whether the current viewport displays a perspective view.

**Home View**

You can define a Home view for a model so you can restore a familiar view when you use the navigation tools.

The Home view is a special view stored with a model that makes it easy to return to a known or familiar view. You can define any view of the model as the Home view. The saved Home view can be applied to the current view by clicking the Home button above the ViewCube tool or from the ViewCube menu.

When you open a drawing that was created in a release earlier than AutoCAD 2008, the extents of a model are used as the default Home view. Drawings created with AutoCAD 2011 for Mac have a Home view defined with a Top/Left/Front orientation. While you can use the Home view to navigate back to a familiar view, you can also use it to generate the thumbnail preview when you save a model instead of using the last saved view.
Quick Reference

Commands
NAVVCUBE

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

Examine Individual Objects with ViewCube

You can examine individual objects of a model using the ViewCube tool.

With the ViewCube tool, you can define the center of a view based on one or more selected objects. Select an object or objects and use the ViewCube tool to reorient the model. The model rotates around the center of the view. Calculate the center of the view by the extents of the selected objects.

Quick Reference

Commands
NAVVCUBE

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

Change the UCS with ViewCube

With the ViewCube tool you can change the current UCS for the model to one of the named UCSs saved with the model or you can define a new UCS.

The UCS menu, located below the ViewCube tool, displays the name of the current UCS in the model. From the menu, you can restore one of the named UCSs saved with the model, switch to WCS, or define a new UCS. With the WCS item on the menu, you can switch the coordinate system from the current UCS to WCS. With the new UCS, you can rotate the current UCS based on one, two, or three points to define a new UCS. When you click New UCS, a new UCS is defined with the default name of Unnamed. To save the new defined UCS with a name, use the Named option in the UCS command.

You can orient the ViewCube tool with the current UCS or WCS. By orienting the ViewCube tool with the current UCS, you know in which direction you are modeling. Orienting the ViewCube tool with the WCS, you can navigate
the model based on the North and Up directions of the model. The settings for controlling the orientation of the ViewCube tool are in the ViewCube Settings dialog box.

![ViewCube Tool](image)

**Quick Reference**

**Commands**

**NAVVCUBE**

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

**UCS**

Manages user coordinate systems.

**System Variables**

**NAVVCUBEORIENT**

Controls whether the ViewCube tool reflects the current UCS or WCS.
Display Multiple Views in Model Space

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

**Set Model Space Viewports**

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

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

Viewports created on the Model layout completely fill the drawing area and do not overlap. As you make changes in one viewport, the others are updated simultaneously. Three model space viewports are shown in the illustration.
You can also create viewports on a named (paper space) layout. You use those viewports, called *layout viewports*, to arrange the views of your drawing on a sheet. You can move and resize layout viewports. By using layout viewports, you have more control over the display; for example, you can freeze certain layers in one layout viewport without affecting the others. For more information about layouts and layout viewports, see *Create Multiple-View Drawing Layouts (Paper Space)* on page 137.

**Use Model Space Viewports**

With model space viewports, you can do the following:

- Pan; zoom; set Snap, Grid, and UCS icon modes; and restore named views.
- Save user coordinate system orientations with individual viewports.
- Draw from one viewport to another when executing a command.
- Name a viewport arrangement so that you can reuse it on the Model layout or insert it on a named layout.

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

**Split and Join Model Space Viewports**

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

**Quick Reference**

**Commands**

**MODEL**

Switches from a named (paper space) layout to the Model layout.

**VPORTS**

Creates multiple viewports in model space or paper space.

**System Variables**

**MAXACTVP**

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

**CTAB**

Returns the name of the current layout in the drawing.

**TILEMODE**

Makes the Model tab or the last layout tab current.
Select and Use the Current Viewport

When you use multiple viewports, one of them is the *current viewport*, which accepts cursor input and view commands.

When a viewport is current, the cursor is displayed as crosshairs rather than an arrow, and the viewport boundary is highlighted. You can change the current viewport at any time except when a View command is in progress.

To make a viewport the current viewport, you click inside it or press Ctrl-R to cycle through the existing viewports.

To draw a line using two model space viewports, you start the line in the current viewport, make another viewport current by clicking within it, and then specify the endpoint of the line in the second viewport. In a large drawing, you can use this method to draw a line from a detail in one corner to a detail in a distant corner.

Quick Reference

**System Variables**

**CVPORT**
- Displays the identification number of the current viewport.

**VIEWCTR**
- Stores the center of view in the current viewport.
VIEWSIZE

Stores the height of the view displayed in the current viewport, measured in drawing units.

**Save and Restore Model Layout Viewport Arrangements**

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

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

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

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

**Quick Reference**

**Commands**

RENAME

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

VPORTS

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

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

With this method, you create the building, mechanical part, or geographic area that you want to represent at full scale (1:1), but you create the text, dimensions, and the title block of the drawing at a scale to match the intended plot scale.

Quick Start for Model Space Drafting

The process of creating and plotting a drawing file in model space is very different from the process used in manual drafting.

In AutoCAD for Mac, there are two distinct working environments that are represented by Model and named layouts.

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

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

For information about using annotative objects and scaling annotations automatically, see Scale Annotations on page 656.

See also:
■ Create Multiple-View Drawing Layouts (Paper Space) on page 137

Quick Reference

Commands
MODEL
Switches from a named (paper space) layout to the Model layout.
RENAMEN
Changes the names assigned to items such as layers and dimension styles.
VPORTS
Creates multiple viewports in model space or paper space.

System Variables
CVPORT
Displays the identification number of the current viewport.
MAXACTVP
Sets the maximum number of viewports that can be active at one time in a layout.
TILEMODE
Makes the Model tab or the last layout tab current.
VIEWCTR
Stores the center of view in the current viewport.
VIEWSIZE
Stores the height of the view displayed in the current viewport, measured in drawing units.
Draw, Scale, and Annotate in Model Space

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

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

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

You can also use objects if you want to scale annotations automatically. For information about using annotative objects and scaling annotations automatically, see Scale Annotations on page 656.

Determine the Unit of Measurement

Before you begin drawing in model space, you determine the unit of measurement (drawing units) that you plan to use. You decide what each unit on the screen represents, such as an inch, a millimeter, a kilometer, or some other unit of measurement. For example, if you are drawing a motor part, you might decide that one drawing unit equals a millimeter. If you are drawing a map, you might decide that one unit equals a kilometer.

Specify the Display Style of Drawing Units

Once you have determined a drawing unit for the drawing, you need to specify the style for displaying the drawing unit, which includes the unit type and precision. For example, a value of 14.5 can be displayed as 14.500, 14-1/2, or 1’2-1/2”.

Specify the display style of drawing units with the UNITS command. The default drawing unit type is decimal.
Set the Scale for Annotations and Blocks

Before you draw, you should set the scale for dimensions, annotations, and blocks in your drawings. Scaling these elements beforehand ensures that they are at the correct size when you plot the final drawing.

You should enter the scale for the following objects:

- **Text.** Set the text height as you create text or by setting a fixed text height in the text style (STYLE).

- **Dimensions.** Set the dimension scale in a dimension style (DIMSTYLE) or with the DIMSCALE system variable.

- **Linetypes.** Set the scale for noncontinuous linetypes with the CELTSCALE and LTSCALE system variables.

- **Hatch patterns.** Set the scale for hatch patterns in the Hatch and Gradient dialog box (HATCH) or with the HPSCALE system variable.

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

You can also use objects if you want to scale annotations automatically. For information about using annotative objects and scaling annotations automatically, see Scale Annotations on page 656.

Determine the Scale Factor for Plotting

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

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

\[
\frac{1/4\text{"}}{12\text{"}} = \frac{1}{48}
\]

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.

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

If you are working in metric units, you might have a sheet size of 210 x 297 mm (A4 size) and a scale factor of 20. You calculate grid limits as follows:

210 x 20 = 4200 mm
297 x 20 = 5900 mm

See also:
- Specify Units and Unit Formats on page 54

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.
INSERT
 Inserts a block or drawing into the current drawing.

LINETYPE
 Loads, sets, and modifies linetypes.

PLOT
 Outputs a drawing to a printer or file.

STYLE
 Creates, modifies, or specifies text styles.

UNITS
 Controls coordinate and angle display formats and precision.

System Variables

CELTSCALE
 Sets the current object linetype scaling factor.

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

HPSSCALE
 Sets the hatch pattern scale factor.

HPSPACE
 Sets the hatch pattern line spacing for user-defined patterns.

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

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

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

LUNITS

Sets linear units.

TEXTSIZE

Sets the default height for new text objects drawn with the current text style.
Create Multiple-View Drawing Layouts (Paper Space)

*Paper space* is a sheet layout environment where you can specify the size of your sheet, add a title block, display multiple views of your model, and create dimensions and notes for your drawing.

**Quick Start for Layouts**

There are two distinct working environments, or “spaces,” in which you can create objects in a drawing. These are represented by the Model and named layouts.

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*.

Working on the Model layout, you draw a model of your subject at 1:1 scale. Working on a named layout, you can create one or more *layout viewports*, dimensions, notes, and a title block to represent a drawing sheet.

Each layout viewport is like a picture frame containing a “photograph” of the model in model space. Each layout viewport contains a view that displays the model at the scale and orientation that you specify. You can also specify which layers are visible in each layout viewport.

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

LAYOUT
  Creates and modifies drawing layouts.

MODEL
  Switches from a named (paper space) layout to the Model layout.

MSPACE
  In a layout, switches from paper space to model space in a layout viewport.

MVIEW
  Creates and controls layout viewports.

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

PSETUPIN
  Imports a user-defined page setup into a new drawing layout.

PSPACE
  In a layout, switches from model space in a viewport to paper space.

VPORTS
  Creates multiple viewports in model space or paper space.

VPLAYER
  Sets layer visibility within viewports.

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

PSLTSCALE
  Controls the linetype scaling of objects displayed in paper space viewports.

TILEMODE
  Makes the Model tab or the last layout tab current.
Understand the Layout Process

When you use a named layout to prepare your drawing for output, you follow a series of steps in a process.

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

There is one Model layout and one or more named layouts in a drawing. A drawing always has at least one named layout.

Before you can use a layout, it must be initialized. A layout does not contain any page setup information before it is initialized. Once initialized, layouts can be drawn upon and output.

Process Summary

When you prepare a layout, you typically step through the following process:

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

You can also use objects if you want to annotate your drawing in model space and scale the annotations automatically. For information about using annotative objects and scaling annotations automatically, see Scale Annotations on page 656.

The other topics in this chapter provide additional detail on how to create, use, and modify layouts and layout viewports.
Quick Reference

LAYOUT
Creates and modifies drawing layouts.

MODEL
Switches from a named (paper space) layout to the Model layout.

MSPACE
In a layout, switches from paper space to model space in a layout viewport.

MVIEW
Creates and controls layout viewports.

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

PLOT
Outputs a drawing to a printer or file.

PSETUPIN
Imports a user-defined page setup into a new drawing layout.

PSPACE
In a layout, switches from model space in a viewport to paper space.

VPLAYER
Sets layer visibility within viewports.

VPMAX
Expands the current layout viewport for editing.

VPMIN
Restores the current layout viewport.

VPORTS
Creates multiple viewports in model space or paper space.

LAYOUTREGENCTL
Specifies how the display list is updated in the Model tab and layout tabs.
MAXACTVP
Sets the maximum number of viewports that can be active at one time in a layout.

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

Work with Model Space and Paper Space

There are several benefits to switching between model space and paper space to perform certain tasks. Use model space for creating and editing your model. Use paper space for composing your drawing sheet and defining views.

Work on the Model Layout

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

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

On the Model layout, 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.

Quick Reference

MODEL
Switches from a named (paper space) layout to the Model layout.

MSPACE
In a layout, switches from paper space to model space in a layout viewport.

PSPACE
In a layout, switches from model space in a viewport to paper space.

LAYOUTREGENCTL
Specifies how the display list is updated in the Model tab and layout tabs.
MAXACTVP
Sets the maximum number of viewports that can be active at one time in a layout.

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

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

In paper space, one unit represents the actual distance on a sheet of paper. The units will be in either millimeters or inches, depending on how you configure your page setup.

On a named layout, you can view and edit paper space objects, such as layout viewports and title blocks. You can also move an object (such as a leader or a title block) from model space to paper space (or vice versa). The crosshairs cursor is active over the entire layout area.

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

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

- Add a new layout with no settings and then specify the settings in the Page Setup Manager.
- Copy a layout and its settings from the current drawing file.
- Import a layout from an existing drawing template (DWT) file or drawing (DWG) file.

Quick Reference

LAYOUT
Creates and modifies drawing layouts.
MODEL
Switches from a named (paper space) layout to the Model layout.

MSPACE
In a layout, switches from paper space to model space in a layout viewport.

MVIEW
Creates and controls layout viewports.

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

PLOT
Outputs a drawing to a printer or file.

PSETUPIN
Imports a user-defined page setup into a new drawing layout.

PSPACE
In a layout, switches from model space in a viewport to paper space.

VPORTS
Creates multiple viewports in model space or paper space.

VPLAYER
Sets layer visibility within viewports.

CTAB
Returns the name of the current layout in the drawing.

CVPORT
Displays the identification number of the current viewport.

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

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

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

Access Model Space from a Layout Viewport
You can access model space from a layout viewport to edit objects, to freeze and thaw layers, and to adjust the view.

After creating viewport objects, you can access model space from a layout viewport to perform the following tasks:

- Create and modify objects in model space inside the layout viewport.
- Pan the view inside the layout viewport and change layer visibility.

The method you use to access model space depends on what you plan to do.

Create and Modify Objects in a Layout Viewport
If you plan to create or modify objects, use the 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 is restored before the Print dialog box is displayed. If you save and close the drawing while a viewport is maximized, the drawing opens with the named layout restored.

If you choose to switch to the Model layout 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 Layers palette, and you can pan the view. To return to paper space, double-click an empty area on the layout outside a viewport. The changes you made are displayed in the viewport.

If you set the scale in the layout viewport before you access model space, you can lock the scale to prevent changes. When the scale is locked, you cannot use ZOOM while you work in model space.

**Quick Reference**

**MODEL**
- Switches from a named (paper space) layout to the Model layout.

**MSPACE**
- In a layout, switches from paper space to model space in a layout viewport.

**PSPACE**
- In a layout, switches from model space in a viewport to paper space.

**VPMAX**
- Expands the current layout viewport for editing.

**VPMIN**
- Restores the current layout viewport.

**VPMAXIMIZEDSTATE**
- Indicates whether the viewport is maximized or not.

**Create and Modify Layout Viewports**

You can create a single layout viewport that fits the entire layout or create multiple layout viewports in the layout. Once you create the viewports, you can change their size, their properties, and also scale and move them as needed.

**NOTE** It is important to create layout viewports on their own layer. When you are ready to output your drawing, you can turn off the layer and output the layout without the boundaries of the layout viewports.

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

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

You can use the MVIEW command to create nonrectangular viewports.

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

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

**NOTE** When you want to suppress the display of the boundary of a layout viewport, you should turn off the layer of the nonrectangular viewport instead of freezing it. If the layer of a nonrectangular layout viewport is frozen, the viewport is not clipped correctly.

Redefine Layout Viewport Boundaries

You can redefine the boundary of a layout viewport by using the VPCLIP command. You can either select an existing object to designate as the new boundary, or specify the points of a new boundary. The new boundary does not clip the old boundary; it redefines it.
A nonrectangular viewport consists of two objects: the viewport itself and the clipping boundary. You can make changes to the viewport, the clipping boundary, or both.

**NOTE** In the Properties Inspector, the default selection for a nonrectangular viewport is Viewport. This is because you are more likely to change the properties of the viewport than of the clipping boundary.

### Resize Layout Viewports

If you want to change the shape or size of a layout viewport, you can use grips to edit the vertices just as you edit any object with grips.

### Quick Reference

- **MVIEW**
  
  Creates and controls layout viewports.

- **PROPERTIES**
  
  Controls properties of existing objects.

- **VPCLIP**
  
  Clips layout viewport objects and reshapes the viewport border.

- **VPORTS**
  
  Creates multiple viewports in model space or paper space.

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

### Control Views in Layout Viewports

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

### Scale Views in Layout Viewports

To scale each displayed view in output accurately, set the scale of each view relative to paper space.
You can change the view scale of the viewport using

- The Properties Inspector
- The XP option of the ZOOM command
- The Viewports Scale on the status bar

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

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

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

When creating a new drawing based on a template, the scales in the template are used in the new drawing. The scales in the user profile are not imported.

**Lock the Scale of Layout Viewports**

Once you set the viewport scale, you cannot zoom within a viewport without changing the viewport scale. By locking the viewport scale first, you can zoom in to view different levels of detail in your viewport without altering the viewport scale.

Scale locking locks the scale that you set for the selected viewport. Once the scale is locked, you can continue to modify the geometry in the viewport without affecting the viewport scale. If you turn a viewport's scale locking on, most of the viewing commands, such as VPOINT, DVIEW, 3DORBIT, PLAN, and VIEW, no longer function in that viewport.

**NOTE** Viewport scale locking is also available for nonrectangular viewports. To lock a nonrectangular viewport, you must perform an extra step in the Properties Inspector to select the viewport object rather than the viewport clipping boundary.

**Annotative Objects and Scaling**

Annotative objects are defined at a paper height instead of a model size and assigned one or more scales. These objects are scaled based on the current annotation scale setting and automatically displayed at the correct size in the
layout or when plotted. The annotation scale controls the size of the annotative objects relative to the model geometry in the drawing.

You can specify the default list of scales available for layout viewports, page layouts, and printing in Default Scale List dialog box.

For more information about annotation scaling, see Scale Annotations on page 656.

Quick Reference

SCALELISTEDIT
Controls the list of scales available for layout viewports, page layouts, and plotting.

MVIEW
Creates and controls layout viewports.

PROPERTIES
Controls properties of existing objects.

VPORTS
Creates multiple viewports in model space or paper space.

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

Control Visibility in Layout Viewports

You can control the visibility of objects in layout viewports using several methods. These methods are useful for emphasizing or hiding different elements of a drawing, and for reducing screen regeneration time.

See also:

■ Display Annotative Objects on page 670

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 Layers palette.

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

**Freeze or Thaw Layers Automatically in New Layout Viewports**

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

**Create New Layers That Are Frozen in All Layout Viewports**

You can create new layers that are frozen in all existing and new layout viewports. Then you can thaw the layers in the viewports you specify. This is a shortcut for creating a new layer that is visible only in a single viewport.
Quick Reference

LAYER
Manages layers and layer properties.

VPLAYER
Sets layer visibility within viewports.

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

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

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

See also:
■ Set Options for Plotted Objects on page 870

Quick Reference

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

PROPERTIES
Controls properties of existing objects.

Turn Layout Viewports On or Off
You can save time by turning some layout viewports off or by limiting the number of active viewports.
Displaying a large number of active layout viewports can affect your system’s performance as the content of each layout viewport regenerates. You can save time by turning some layout viewports off or by limiting the number of active viewports. The following illustration shows the effects of turning off two layout viewports.

New layout viewports are turned on by default. If you turn off the layout viewports you aren’t using, you can copy layout viewports without waiting for each one to regenerate.

If you don’t want to plot a layout viewport, you can turn the layout viewport off.

**Quick Reference**

**MVIEW**
- Creates and controls layout viewports.

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

**PROPERTIES**
- Controls properties of existing objects.

**VPORTS**
- Creates multiple viewports in model space or paper space.

**MAXACTVP**
- Sets the maximum number of viewports that can be active at one time in a layout.
Scale Linetypes in Layout Viewports

You can scale linetypes in paper space either based on the drawing units of the space in which the object was created or based on the paper space units.

You can set the PSLTSCALE system variable to maintain the same linetype scaling for objects displayed at different zoom factors in a layout and in a layout viewport. For example, with PSLTSCALE set to 1 (default), set the current linetype to dashed, and then draw a line in a paper space layout.

In the layout, create a viewport with a zoom factor of 1x, make that layout viewport current, and then draw a line using the same dashed linetype. The dashed lines should appear to be the same. If you change the viewport zoom factor to 2x, the linetype scaling for the dashed line in the layout and the dashed line in the layout viewport will be the same, regardless of the difference in the zoom factor.

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

See also:

■ Set the Lineweight Scale for a Layout on page 853

Quick Reference

LINETYPE

Loads, sets, and modifies linetypes.

PSLTSCALE

Controls the linetype scaling of objects displayed in paper space viewports.
Align Views in Layout Viewports

You can arrange the elements of your drawing by aligning the view in one layout viewport with the view in another viewport.

For angled, horizontal, and vertical alignments, you can move each layout viewport relative to distances defined by the model-space geometry displayed.

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

Quick Reference

MOVE

Moves objects a specified distance in a specified direction.
MVSETUP
Sets up the specifications of a drawing.

UCS
Manages user coordinate systems.

UCSICON
Controls the visibility and placement of the UCS icon.

UCSMAN
Manages defined user coordinate systems.

UCSICON
Controls the visibility and placement of the UCS icon.

UCSVP
Determines whether the UCS in viewports remains fixed or changes to reflect the UCS of the current viewport.

**Rotate Views in Layout Viewports**

You can rotate an entire view within a layout viewport with the VPROTATEASSOC system variable.

When VPROTATEASSOC is set to 1, the view within a viewport is rotated with the viewport. When VPROTATEASSOC is set to 0, the view remains when the viewport is rotated.

You can also rotate an entire view within a layout viewport by changing the UCS and using the PLAN command.

With the UCS command, you can rotate the $XY$ plane at any angle around the $Z$ axis. When you enter the PLAN command, the view rotates to match the orientation of the $XY$ plane.
Another way is to use the Align and then Rotate View options in the MVSETUP command.

**NOTE** The ROTATE command rotates individual objects only and should not be used to try to rotate a view.

### Quick Reference

**MVSETUP**
- Sets up the specifications of a drawing.

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

**UCS**
- Manages user coordinate systems.

**UCSICON**
- Controls the visibility and placement of the UCS icon.

**UCSMAN**
- Manages defined user coordinate systems.

**UCSICON**
- Controls the visibility and placement of the UCS icon.

**UCSVP**
- Determines whether the UCS in viewports remains fixed or changes to reflect the UCS of the current viewport.

**VIEWTWIST**
- Stores the view rotation angle for the current viewport measured relative to the WCS.

**VPROTATEASSOC**
- Controls whether the view within a viewport is rotated with the viewport when the viewport is rotated.
Reuse Layouts and Layout Settings

When you create a layout, you can choose to apply the information from an existing template.

A layout template is a layout imported from a DWG or DWT file. When you create a layout, you can choose to apply the information from an existing template. The program has sample layout templates to use when you design a new layout environment. The paper space objects and page setup in the existing template are used in the new layout. Thus, the layout objects, including any viewport objects, are displayed in paper space. You can keep any of the existing objects from the template you import, or you can delete the objects. No model space objects are imported.

The layout templates are identified with a .dwt file extension. However, a layout template or layout from any drawing or drawing template can be imported into the current drawing.

Save a Layout Template

Any drawing can be saved as a drawing template (DWT file), including all of the objects and layout settings. You can save a layout to a new DWT file by choosing the Save As option of the LAYOUT command. The template file is saved in the drawing template file folder as defined in the Application tab (Application Preferences dialog box). The layout template has a .dwt or .dwg extension like a drawing template or drawing file, but it contains little information not essential to the layout.

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

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

**LAYOUT**
- Creates and modifies drawing layouts.

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

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

**TDCREATE**
- Stores the local time and date the drawing was created.

**TDUCREATE**
- Stores the universal time and date that the drawing was created.

**TDUPDATE**
- Stores the local time and date of the last update/save.

**TDUUPDATE**
- Stores the universal time and date of the last update or save.
Create and Modify Objects
Control the Properties of Objects

You can organize objects in your drawing and control how they are displayed and plotted by changing their properties, which include layer, linetype, color, lineweight, transparency, and plot style.

Work with Object Properties

You can change the object properties in your drawing by using the Properties Inspector palette.

Overview of Object Properties

Every object you draw has properties. Some properties are general and apply to most objects; for example, layer, color, linetype, transparency, and plot style. Other properties are object-specific; for example, the properties of a circle include radius and area, and the properties of a line include length and angle.

Most general properties can be assigned to an object by layer or can be assigned to an object directly.

- When a property is set to the value BYLAYER, the object is assigned the same value as the layer on which it is drawn. For example, if a line drawn on layer 0 is assigned the color BYLAYER, and layer 0 is assigned the color Red, the line is red.

- When a property is set to a specific value, that value overrides the value set for the layer. For example, if a line drawn on layer 0 is assigned the color Blue, and layer 0 is assigned the color Red, the line is blue.
See also:
- Control the Color and Linetype Properties in Blocks on page 416

Quick Reference

CHPROP
- Changes the properties of an object.

PROPERTIES
- Controls properties of existing objects.

PROPERTIESCLOSE
- Closes the Properties Inspector palette.

CECOLOR
- Sets the color of new objects.

CELTYPE
- Sets the linetype of new objects.

CELWEIGHT
- Sets the linewidth of new objects.

CETRANSPARENCY
- Sets the transparency level for new objects.

CLAYER
- Sets the current layer.

CPlotSTYLE
- Controls the current plot style for new objects.

PLOTTRANSPARENCYOVERRIDE
- Controls whether object transparency is plotted.

TRANSPARENCYDISPLAY
- Controls whether the object transparency is displayed.
Display and Change the Properties of Objects

You can display and change the current properties for any object in your drawing.

Use the Properties Inspector Palette

The Properties Inspector 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 Inspector palette displays only those properties common to all objects in the selection set.
- When no objects are selected, the Properties Inspector 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 Inspector palette when the DBLCLKEDIT system variable is set to On (the default). The exceptions include blocks and attributes, hatch, text, multilines, and xrefs. If you double-click any of these objects, an object-specific dialog box is displayed instead.

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

Change Object Property or ByBlock Settings to ByLayer

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

See also:

- Control the Color and Linetype Properties in Blocks on page 416
- Set Interface Options on page 43
Quick Reference

DSETTINGS
Sets grid and snap, polar and object snap tracking, object snap modes, and Dynamic Input.

ID
Displays the UCS coordinate values of a specified location.

LIST
Displays property data for selected objects.

PROPERTIES
Controls properties of existing objects.

PROPERTIESCLOSE
Closes the Properties Inspector palette.

CETRANSPARENCY
Sets the transparency level for new objects.

DBLCLKEDIT
Controls the double click editing behavior in the drawing area.

LUPREC
Sets the display precision for linear units and coordinates.

OPMSTATE
Indicates whether the Properties Inspector palette is open or closed.

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

TRANSPARENCYDISPLAY
Controls whether the object transparency is displayed.

Copy Properties Between Objects

You can copy some or all properties of one object to other objects using Match Properties.
The types of properties that can be copied include, but are not limited to, color, layer, linetype, linetype scale, linewidth, plot style, transparency, viewport property overrides, and 3D thickness.

By default, all applicable properties are automatically copied from the first object you selected to the other objects. If you don’t want a specific property or properties to be copied, use the Settings option to suppress the copying of that property. You can choose the Settings option at any time during the command.

Quick Reference

MATCHPROP

Applies the properties of a selected object to other objects.

Work with Layers

Layers are like transparent overlays on which you organize and group objects in a drawing.

Overview of Layers

Layers are used to group information in a drawing by function and to enforce linetype, color, and other standards.

Layers are the equivalent of the overlays used in paper-based drafting. Layers are the primary organizational tool used in drawing. Use layers to group information by function and to enforce linetype, color, and other standards.
By creating layers, you can associate similar types of objects by assigning them to the same layer. For example, you can put construction lines, text, dimensions, and title blocks on separate layers. You can then control the following:

- Whether objects on a layer are visible or dimmed in any viewports
- Whether and how objects are plotted
- What color is assigned to all objects on a layer
- What default linetype and lineweight are assigned to all objects on a layer
- Whether objects on a layer can be modified
- Whether objects display with different layer properties in individual layout viewports

Every drawing includes a layer named 0. Layer 0 cannot be deleted or renamed. It has two purposes:

- Ensure that every drawing includes at least one layer
- Provide a special layer that relates to controlling colors in blocks

**NOTE** It is recommended that you create several new layers with which to organize your drawing rather than create your entire drawing on layer 0.

**Quick Reference**

**LAYER**

Manages layers and layer properties.
Use Layers to Manage Complexity

You can use layers to control the visibility of objects and to assign properties to objects. Layers can be locked to prevent objects from being modified.

You can reduce the visual complexity of a drawing and improve display performance by controlling how objects are displayed or plotted. For example, you can use layers to control the properties and visibility of similar objects, such as electrical parts or dimensions. Also, you can lock a layer to prevent objects on that layer from being accidentally selected and modified.

Control the Visibility of Objects on a Layer

You can make drawing layers invisible either by turning them off or by freezing them. Turning off or freezing layers is useful if you need an unobstructed view when working in detail on a particular layer or set of layers or if you don’t want to plot details such as reference lines. Whether you choose to freeze layers or turn them off depends on how you work and on the size of your drawing.

■ **On/Off.** Objects on turned-off layers are invisible, but they still hide objects when you use HIDE. When you turn layers on and off, the drawing is not regenerated.

■ **Freeze/Thaw.** Objects on frozen layers are invisible and do not hide other objects. In large drawings, freezing unneeded layers speeds up operations involving display and regeneration. Thawing one or more layers may cause the drawing to be regenerated. Freezing and thawing layers takes more time than turning layers on and off.

In a layout, you can freeze layers in individual layout viewports.

**NOTE** Instead of turning off or freezing a layer, you can fade the layer by locking it. See “Lock the Objects on a Layer” below.

Control Transparency on Layers

Set the transparency of layers and layout viewports to enhance drawings by reducing the visibility of all objects on specific layers as needed. Set layer (or layout viewport) transparency in the Layers palette.

After you apply transparency to a layer, all objects added to that layer are created at the same level of transparency. The transparency property for all objects on the layer is set to ByLayer.
Assign a Default Color and Linetype to a Layer

Each layer has associated properties such as color, linetype, and transparency that are assumed by all objects on that layer when the setting is ByLayer.

For example, if the Properties Inspector palette set to BYLAYER when no object is selected, the color of new objects is determined by the color setting for the layer in the Layers palette.

If you set a specific color to the Properties Inspector palette when no objects are selected, that color is used for all new objects, overriding the default color for the current layer. The same is true for Linetype, Lineweight, Transparency, and Plot Style properties on the Properties Inspector palette.

The BYBLOCK setting should be used only for creating blocks. See Control the Color and Linetype Properties in Blocks on page 416.

Override Layer Properties in a Layout Viewport

Some layer properties can be changed using overrides on a viewport basis in layouts. Using layer property overrides is an efficient way to display objects with different property settings for color, linetype, lineweight, transparency, and plot style. Layer property overrides are applied to the current layout viewport.

For example, if you want objects on the Electrical layer to display prominently in one of two layout viewports, you set a Color override on the Electrical layer for each of the two viewports. By setting the color red for one viewport and gray for the other, you easily accomplish this objective without changing the global color property assigned to the layer. See Override Layer Properties in Viewports on page 173 for more information.

Lock the Objects on a Layer

When a layer is locked, none of the objects on that layer can be modified until you unlock the layer. Locking layers reduces the possibility of modifying objects accidentally. You can still apply object snaps to objects on a locked layer and perform other operations that do not modify those objects.

You can fade the objects on locked layers to make them appear more faint than other objects. This serves two purposes:

- You can easily see what objects are on locked layers.
- You can reduce the visual complexity of a drawing but still maintain visual reference and object snapping capabilities to those objects.
The LAYLOCKFADECTL system variable controls the fading applied to locked layers. Locked layers that are faded are plotted normally.

When you lock a layer that contains transparent objects, the visibility of those objects is further reduced by the specified locked layer fading value.

**NOTE** Grips are not displayed on objects that are on locked layers.

**Quick Reference**

**LAYER**
- Manages layers and layer properties.

**LAYFRZ**
- Freezes the layer of selected objects.

**LAYISO**
- Hides or locks all layers except those of the selected objects.

**LAYMCH**
- Changes the layer of a selected object to match the destination layer.

**LAYMCUR**
- Sets the current layer to that of a selected object.

**LAYOFF**
- Turns off the layer of a selected object.

**LAYUNISO**
- Restores all layers that were hidden or locked with the LAYISO command.

**LAYLOCKFADECTL**
- Controls the amount of fading for objects on locked layers.

**Create and Name Layers**

You can create and name a new layer for each conceptual grouping (such as walls or dimensions) and assign common properties to each layer.

By organizing objects into layers, you can control the visibility and object properties of a large number of objects separately for each layer and make changes quickly.
NOTE The number of layers that you can create in a drawing and the number of objects that you can create on each layer are practically unlimited.

Choose Layer Names Carefully

A layer name can include up to 255 characters (double-byte or alphanumeric): letters, numbers, spaces, and several special characters. Layer names cannot include the following characters:
< > \ “ ; ? * | = ’

In many cases, the layer names you choose are dictated by corporate, industry, or client standards.

The Layers palette sorts layers alphabetically by name. If you organize your own layer scheme, choose layer names carefully. Use common prefixes to name layers with related drawing components, this makes it easier to locate and manipulate groups of layers at a time.

NOTE If you consistently use a specific layering scheme, you can set up a drawing template with layers, linetypes, and colors already assigned. For more information about creating templates, see Use a Drawing Template File on page 58.

Select a Layer to Draw On

As you draw, newly created objects are placed on the current layer. The current layer may be the default layer (0) or a layer you create and name yourself. You switch from one layer to another by making a different layer current; any subsequent objects you create are associated with the new current layer and use its color, linetype, and other properties. You cannot make a layer the current layer if it is frozen or if it is an xref-dependent layer.

Remove Layers

You can remove unused layers from your drawing with PURGE or by deleting the layer from the Layers palette. 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.
Quick Reference

LAYER
Manages layers and layer properties.

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

CLAYER
Sets the current layer.

Change Layer Settings and Layer Properties

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

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

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

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

You change layer properties in the Layers palette. Click the icons to change settings, and rename layers or change descriptions.

Undo Changes to Layer Settings

You can use Layer Previous to undo changes you make to layer settings. For example, if you freeze several layers and change some of the geometry in a drawing, and then want to thaw the frozen layers, you can do this with a single command without affecting the geometry changes. In another example, if you changed the color and linetype of several layers but later decide you prefer the old properties, you can use Layer Previous to undo the changes and restore the original layer settings.
When you use Layer Previous, it undoes the most recent layer change or set of changes made. Every change you make to layer settings is tracked and can be undone with Layer Previous. You can use LAYERPMODE to suspend layer property tracking when you don't need it, such as when you run large scripts. There is a modest performance gain in turning off Layer Previous tracking.

Layer Previous does not undo the following changes:

- **Renamed layers.** If you rename a layer and change its properties, Layer Previous restores the original properties but not the original layer name.
- **Deleted layers.** If you delete or purge a layer, using Layer Previous does not restore it.
- **Added layers.** If you add a new layer to a drawing, using Layer Previous does not remove it.

**Quick Reference**

**CHANGE**
Changes the properties of existing objects.

**CHPROP**
Changes the properties of an object.

**LAYER**
Manages layers and layer properties.

**LAYERP**
Undoes the last change or set of changes made to layer settings.

**LAYERPMODE**
Turns on and off the tracking of changes made to layer settings.

**PROPERTIES**
Controls properties of existing objects.

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

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

Sets the current layer.

Override Layer Properties in Viewports

You can display objects differently by setting property overrides for color, linetype, lineweight, transparency, and plot style and apply them to individual layout viewports.

Using property overrides is an efficient way for displaying objects with different property settings in individual viewports without changing their ByLayer or ByBlock properties. For example, objects can be made to display more prominently by changing their color. Because layer property overrides do not change the layer's global properties, you can have objects display differently in various viewports without having to create duplicate geometry or use xrefs that have different layer settings.

Property override settings for color and lineweight were set on the Wiring layer for the viewport on the left. Notice the wiring is a different color and lineweight than in the right viewport.

When the Layers palette is accessed from a layout tab, five columns for layer property overrides are displayed

- VP Color
- VP Linetype

Override Layer Properties in Viewports | 173
- VP Lineweight
- VP Transparency
- VP Plot Style (available only in named-plot style drawings)

If you do not want to display or plot property overrides, set the VPLAYEROVERRIDESMODE system variable to 0. Objects will display and plot with their global layer properties.

**NOTE** Property overrides can still be set even when VPLAYEROVERRIDESMODE is set to 0.

Property overrides that are on xref layers are not retained when the VISRETAIN system variable is set to 0.

**Viewport Property Overrides and Visual Styles**

Layer property overrides for color, linetype, and lineweights are displayed in viewports regardless of the visual style that is current. Although plot style overrides can be set when the visual style is set to Conceptual or Realistic style, they are not displayed or plotted.

**Identify Viewports with Layer Property Overrides**

Use the VPLAYEROVERRIDES system variable to check if the current viewport contains layer property overrides. When VPLAYEROVERRIDES is equal to 1, the viewport contains overrides.

You can also use the Properties Inspector palette to determine if a viewport contains overrides. The Properties Inspector palette displays a Layer Property Overrides field. The value that displays is the same as the setting for VPLAYEROVERRIDES.

**Remove Layer Property Overrides**

When you right-click a layer in the Layers palette, a shortcut menu is displayed that lists options for removing property overrides. You can remove

- A single property override from the selected layer for the selected viewport or for all selected viewports
- All property overrides from the selected layer for the selected viewport or for all selected viewports
- All property overrides from all layers in the selected viewport or for all selected viewports

**NOTE** Another method for removing property overrides is to use the shortcut menu when you right-click the border of the selected viewport or viewports. You can remove viewport overrides for all layers for that viewport.

**Quick Reference**

**CHPROP**
Changes the properties of an object.

**LAYER**
Manages layers and layer properties.

**LAYERP**
Undoes the last change or set of changes made to layer settings.

**LAYERMODE**
Turns on and off the tracking of changes made to layer settings.

**PROPERTIES**
Controls properties of existing objects.

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

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

**CLAYER**
Sets the current layer.

**VPLAYEROVERRIDES**
Indicates if there are any layers with viewport (VP) property overrides for the current layout viewport.

**VPLAYEROVERRIDESMODE**
Controls whether layer property overrides for layout viewports are displayed and plotted.
Work with Colors

Color helps to group objects visually. You can assign colors to objects by layer or individually.

Set the Current Color

You can use color to help you identify objects visually. You can assign the color of an object either by layer or by specifying its color explicitly, independent of layer.

Assigning colors by layer makes it easy to identify each layer within your drawing. Assigning colors explicitly provides additional distinctions between objects on the same layer. Color is also used as a way to indicate lineweight for color-dependent plotting.

You can use a variety of color palettes when assigning color to objects, including

- AutoCAD for Mac 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 for Mac. 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 Applications tab in the Application Preferences dialog box. Once a color book is loaded, you can select colors from the color book and apply them to objects in your drawings.

All objects are created using the current color, which is displayed in the Color property of the Properties Inspector palette when no object is selected. You can also set the current color with the Color Palette 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.

Quick Reference

COLOR
  Sets the color for new objects.

CECOLOR
  Sets the color of new objects.

Change the Color of an Object

You can change the color of an object by reassigning it to another layer, by changing the color of the layer the object is on, or by specifying a color for the object explicitly.
You have three choices for changing the color of an object:

■ Reassign the object to another layer with a different color. If an object's color is set to BYLAYER, and you reassign the object to a different layer, it acquires its color from the new layer.

■ Change the color assigned to the layer that the object is on. If an object's color is set to BYLAYER, it acquires the color of its layer. When you change the color assigned to a layer, all objects on that layer assigned the BYLAYER color are updated automatically.

■ Specify a color for an object to override the layer's color. You can specify the color of each object explicitly. If you want to override the layer-determined color of an object with a different one, change an existing object's color from BYLAYER to a specific color, such as red.

If you want to set a specific color for all subsequently created objects, change the Color property on the Properties Inspector palette when no objects are selected from BYLAYER to a specific color.

See also:

■ Override Layer Properties in Viewports on page 173

Quick Reference

CHANGE
  Changes the properties of existing objects.

CHPROP
  Changes the properties of an object.

COLOR
  Sets the color for new objects.

PROPERTIES
  Controls properties of existing objects.

CECOLOR
  Sets the color of new objects.
**Use Color Books**

When assigning colors to objects, you can choose colors from color books that are loaded on your system.

You can choose from a wide range of custom colors when using color books. Color books include third-party or user-defined files that contain named color swatches. These colors can be used to enhance presentation drawings as well as to optimize the variety of color used in your drawings. You can apply color book colors to objects in your drawings by using the Color Books tab in the Color Palette 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 Color Palette dialog box, you must first copy your color book files to a specified color book location.

On the Application tab of the Application Preferences 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 Color Palette 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.

**Quick Reference**

**CHANGE**

Changes the properties of existing objects.

**CHPROP**

Changes the properties of an object.
COLOR
Sets the color for new objects.

PROPERTIES
Controls properties of existing objects.

CECOLOR
Sets the color of new objects.

Work with Linetypes
You can use linetypes to distinguish objects from one another visually and make your drawing easier to read.

Overview of Linetypes
A linetype is a repeating pattern of dashes, dots, and blank spaces displayed in a line or a curve. You assign linetypes to objects either by layer or by specifying the linetype explicitly, independent of layers.

In addition to choosing a linetype, you can set its scale to control the size of the dashes and spaces, and you can create your own custom linetypes.

NOTE These linetypes should not be confused with the hardware linetypes provided by some plotters. The two types of dashed lines produce similar results. Do not use both types at the same time, however, because the results can be unpredictable.

Some linetype definitions include text and symbols.

---

You can define a custom linetype that will orient the imbedded text to keep it readable automatically.
For more information about controlling text in linetypes, see Text in Custom Linetypes.

See also:
- “Custom Linetypes” in the *Customization Guide*

**Quick Reference**

**LINETYPE**

Loads, sets, and modifies linetypes.

**REVERSE**

Reverses the vertices of selected lines, polylines, splines, and helixes, which is useful for linetypes with included text, or wide polylines with differing beginning and ending widths.

**Load Linetypes**

At the start of a project, you load the linetypes that are required for the project so that they are available when you need them.

If you want to know what linetypes are already available, you can display a list of linetypes that are loaded in the drawing or stored in an LIN (linetype definition) file.

This program includes the linetype definition files *acad.lin* and *acadiso.lin*. Which linetype file is appropriate depends on whether you use imperial or metric measurements.

- For imperial units, use the *acad.lin* file.
- For metric measurements, use the *acadiso.lin* file.

Both linetype definition files contain several complex linetypes.

If you select a linetype whose name begins with ACAD_ISO, you can use the ISO pen-width option when you plot.

You can remove unreferenced linetype information with PURGE or by deleting the linetype from the Linetype Manager. BYBLOCK, BYLAYER, and CONTINUOUS linetypes cannot be removed.
Quick Reference

LINETYPE
 Loads, sets, and modifies linetypes.

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

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

MEASUREINIT
 Controls whether a drawing you start from scratch uses imperial or metric default settings.

MEASUREMENT
 Controls whether the current drawing uses imperial or metric hatch pattern and linetype files.

Set the Current Linetype

All objects are created using the current linetype.
You can set the current linetype with the:

- Linetype property on the Properties Inspector palette
- Linetype Manager

If the current linetype is set to BYLAYER, objects are created with the linetype assigned to the current layer.

If the current linetype is set to BYBLOCK, objects are created using the CONTINUOUS linetype until the objects are grouped into a block. When the block is inserted into the drawing, those objects acquire the current linetype setting.

If you do not want the current linetype to be the linetype assigned to the current layer, you can specify a different linetype explicitly.

The program does not display the linetype of certain objects: text, points, viewports, hatches, and blocks.
Quick Reference

LINETYPE
Loads, sets, and modifies linetypes.

CELTYPE
Sets the linetype of new objects.

Change the Linetype of an Object

You can change the linetype of an object by reassigning it to another layer, by changing the linetype of the layer the object is on, or by specifying a linetype for the object explicitly.

You have three choices for changing the linetype of an object:

- Reassign the object to another layer with a different linetype. If an object’s linetype is set to BYLAYER, and you reassign the object to a different layer, it acquires its linetype from the new layer.
- Change the linetype assigned to the layer that the object is on. If an object’s linetype is set to BYLAYER, it acquires the linetype of its layer. When you change the linetype assigned to a layer, all objects on that layer assigned the BYLAYER linetype are updated automatically.
- Specify a linetype for an object to override the layer’s linetype. You can specify the linetype of each object explicitly. If you want to override the layer-determined linetype of an object with a different one, change an existing object’s linetype from BYLAYER to a specific linetype, such as DASHED.

If you want to set a specific linetype for all subsequently created objects, change the Linetype property on the Properties Inspector palette when no objects are selected from BYLAYER to a specific linetype.
See also:

- Override Layer Properties in Viewports on page 173

Quick Reference

CHANGE
Changes the properties of existing objects.

CHPROP
Changes the properties of an object.

LAYER
Manages layers and layer properties.

LINETYPE
Loads, sets, and modifies linetypes.

PROPERTIES
Controls properties of existing objects.

Control Linetype Scale

You can use the same linetype at different scales by changing the linetype scale factor either globally or individually for each object.

By default, both global and individual linetype scales are set to 1.0. The smaller the scale, the more repetitions of the pattern are generated per drawing unit. For example, with a setting of 0.5, two repetitions of the pattern in the linetype definition are displayed for each drawing unit. Short line segments that cannot display one full linetype pattern are displayed as continuous. You can use a smaller linetype scale for lines that are too short to display even one dash sequence.

The Linetype Manager displays the Global Scale Factor and Current Object Scale.

- The Global Scale Factor value controls the LTSCALE system variable, which changes the linetype scale globally for both new and existing objects.

- The Current Object Scale value controls the CELTSCALE system variable, which sets the linetype scale for new objects.
The CELTSCALE value is multiplied by the LTSCALE value to get the displayed linetype scale. You can easily change linetype scales in your drawing either individually or globally.

In a layout, you can adjust the scaling of linetypes in different viewports with PSLTSCALE.

**Quick Reference**

LINETYPE
   Loads, sets, and modifies linetypes.

CELTSCALE
   Sets the current object linetype scaling factor.

LTSCALE

PSLTSCALE
   Controls the linetype scaling of objects displayed in paper space viewports.

**Display Linetypes on Short Segments and Polylines**

You can center the pattern of a linetype on each segment of a polyline, and you can control how the linetype is displayed on short segments.

If a line is too short to hold even one dash sequence, the result is a continuous line between the endpoints, as shown below.

You can accommodate short segments by using a smaller value for their individual linetype scales. For more information, see Control Linetype Scale on page 184.

For polylines, you can specify whether a linetype pattern is centered on each segment or is continuous across vertices throughout the entire length of the polyline. You do this by setting the PLINEGEN system variable.
Quick Reference

PROPERTIES
Controls properties of existing objects.

PLINEGEN
Sets how linetype patterns generate around the vertices of a 2D polyline.

Control Lineweights
You can control the thickness of an object’s lines in both the drawing display and plotting.

Overview of Lineweights
Lineweights are width values that are assigned to graphical objects as well as some types of text.

Using lineweights, you can create heavy and thin lines to show cuts in sections, depth in elevations, dimension lines and tick marks, and differences in details. For example, by assigning varying lineweights to different layers, you can easily differentiate between new, existing, and demolition construction. Lineweights are not displayed unless the Show/Hide Lineweight button on the status bar is selected.

TrueType fonts, raster images, points, and solid fills (2D solids) cannot display lineweight. Wide polylines show lineweights only when displayed outside of the plan view. You can export drawings to other applications or cut objects to the Clipboard and retain lineweight information.

In model space, lineweights are displayed in pixels and do not change when zoomed in or out. Thus, you should not use lineweights to represent the exact width of an object in model space. For example, if you want to draw an object...
with a real-world width of 0.5 inches, do not use a lineweight; instead, use a polyline with a width of 0.5 inches to represent the object.

**Lineweight Scale in Drawings**

Objects with a lineweight are plotted with the exact width of the assigned lineweight value. The standard settings for these values include BYLAYER, BYBLOCK, and Default. They are displayed in either inches or millimeters, with millimeters being the default. All layers are initially set to 0.25 mm, controlled by the LWDEFAULT system variable.

A lineweight value of 0.025 mm or less is displayed as one pixel in model space and is plotted at the thinnest lineweight available on the specified plotting device. Lineweight values that you enter at the Command prompt are rounded to the nearest predefined value.

You set the lineweight units and the default value in the Lineweight Settings dialog box. You can access the Lineweight Settings dialog box by using the LWEIGHT command, by right-clicking the Show/Hide Lineweight button on the status bar and choosing Settings.

**See also:**

- Draw Polylines on page 262

**Quick Reference**

LWEIGHT

Sets the current lineweight, lineweight display options, and lineweight units.

PLINE

Creates a 2D polyline.

LWDEFAULT

Sets the value for the default lineweight.

LWDISPLAY

Controls whether the lineweights of objects are displayed.

LWUNITS

Controls whether lineweight units are displayed in inches or millimeters.
Display Lineweights

Lineweights can be turned on and off in a drawing, and are displayed differently in model space than in a paper space layout.

- In model space, a 0-value lineweight is displayed as one pixel, and other lineweights use a pixel width proportional to their real-unit value.
- In a paper space layout, lineweights are displayed in the exact plotting width.

Regeneration time increases with lineweights that are represented by more than one pixel. Turn off the display of lineweights to optimize performance of the program.

You can turn the display of lineweights on or off by clicking Show/Hide Lineweight button on the status bar. This setting does not affect the plotting of lineweights.

Display Lineweights in Model Space

Lineweight display in model space does not change with the zoom factor. For example, a lineweight value that is represented by a width of four pixels is always displayed using four pixels regardless of how far you zoom in. If you want the lineweights on objects to appear thicker or thinner on the Model layout, use LWEIGHT to set their display scale. Changing the display scale does not affect the lineweight plotting value.

In model space, weighted lines that are joined form a beveled joint with no end caps. You can use plot styles to apply different joins and endcap styles to objects with lineweights.

NOTE Different styles of endcaps and joins of objects with lineweight are displayed only in a full preview using PREVIEW or PLOT.

Display Lineweights in Layouts

In layouts and plot preview, lineweights are displayed in real-world units, and lineweight display changes with the zoom factor. You can control lineweight plotting and scaling in your drawing in the Print dialog box or the Page Setup Dialog Box.
Quick Reference

LAYER
Manages layers and layer properties.

LWEIGHT
Sets the current lineweight, lineweight display options, and lineweight units.

PEDIT
Edits polylines and 3D polygon meshes.

PLINE
Creates a 2D polyline.

PLOT
Outputs a drawing to a printer or file.

LWDEFAULT
Sets the value for the default lineweight.

LWDISPLAY
Controls whether the lineweights of objects are displayed.

LWUNITS
Controls whether lineweight units are displayed in inches or millimeters.

PLINEWID
Stores the default polyline width.

Set the Current Lineweight

The current lineweight is the lineweight used for any objects you draw until you make another lineweight current.

All objects are created using the current lineweight. You can set the current lineweight with the:

- Lineweight property on the Properties Inspector palette
- Lineweight Settings dialog box
- CELWEIGHT system variable
If the current lineweight is set to BYLAYER, objects are created with the lineweight assigned to the current layer.

If the current lineweight is set to BYBLOCK, objects are created using the default lineweight setting until the objects are grouped into a block. When the block is inserted into the drawing, it acquires the current lineweight setting.

If you do not want the current lineweight to be the lineweight assigned to the current layer, you can specify a different lineweight explicitly.

Objects in drawings created in an earlier release of AutoCAD for Mac 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.

**Quick Reference**

**LAYER**
- Manages layers and layer properties.

**LWEIGHT**
- Sets the current lineweight, lineweight display options, and lineweight units.

**PEDIT**
- Edits polylines and 3D polygon meshes.

**PLINE**
- Creates a 2D polyline.

**PLOT**
- Outputs a drawing to a printer or file.

**LWDEFAULT**
- Sets the value for the default lineweight.

**LWDISPLAY**
- Controls whether the lineweights of objects are displayed.

**LWUNITS**
- Controls whether lineweight units are displayed in inches or millimeters.

**PLINEWID**
- Stores the default polyline width.
Change the Lineweight of an Object

You can change the lineweight of an object by reassigning it to another layer, by changing the lineweight of the layer the object is on, or by specifying a lineweight for the object explicitly.

You have three choices for changing the lineweight of an object:

- Reassign the object to another layer with a different lineweight. If an object’s lineweight is set to BYLAYER, and you reassign the object to a different layer, it acquires its lineweight from the new layer.

- Change the lineweight assigned to the layer that the object is on. If an object’s lineweight is set to BYLAYER, it acquires the lineweight of its layer. When you change the lineweight assigned to a layer, all objects on that layer assigned the BYLAYER lineweight are updated automatically.

- Specify a lineweight for an object to override the layer’s lineweight. You can specify the lineweight of each object explicitly. If you want to override the layer-determined lineweight of an object with a different one, change an existing object’s lineweight from BYLAYER to a specific lineweight.

If you want to set a specific lineweight for all subsequently created objects, change the Lineweight property on the Properties Inspector palette when no objects are selected from BYLAYER to a specific lineweight.

See also:

- Override Layer Properties in Viewports on page 173

Quick Reference

LAYER
Manages layers and layer properties.

LWEIGHT
Sets the current lineweight, lineweight display options, and lineweight units.

PEDIT
Edits polylines and 3D polygon meshes.

PLINE
Creates a 2D polyline.
PLOT
  Outputs a drawing to a printer or file.
LWDEFAULT
  Sets the value for the default lineweight.
LWDISPLAY
  Controls whether the lineweights of objects are displayed.
LWUNITS
  Controls whether lineweight units are displayed in inches or millimeters.
PLINEWID
  Stores the default polyline width.

Control the Display Properties of Certain Objects
You can control how overlapping objects and certain other objects are displayed and plotted.

Control the Display of Polylines, Hatches, Gradient Fills, Lineweights, and Text
You can simplify the display of certain kinds of objects in order to speed performance.

Display performance is improved when wide polylines and donuts, solid-filled polygons (two-dimensional solids), hatches, gradient fills, and text are displayed in simplified form. Simplified display also increases the speed of creating test plots.

Turn Off Solid Fill
When you turn off Fill mode, wide polylines, solid-filled polygons, gradient fill, and hatches are displayed in outline form. Except for patterned hatches and gradient fills, solid fill is automatically turned off for hidden view and nonplan views in three dimensions.
Use Quick Text

When you turn on Quick Text mode in drawings that contain a lot of text using complex fonts, only a rectangular frame defining the text is displayed or plotted.

Turn Off Lineweights

Any lineweight width that is represented by more than one pixel may slow down performance. If you want to improve display performance, turn lineweights off. You can turn lineweights on and off by choosing the Show/Hide Lineweight button on the status bar. Lineweights are always plotted at their real-world value whether their display is turned on or off.

Update the Display

New objects automatically use the current settings for displays of solid fill and text. Except for lineweights, to update the display of existing objects using these settings, you must use REGEN.

See also:

- Use Layers to Manage Complexity on page 167
- Display Lineweights on page 188
- Use TrueType Fonts on page 741
Quick Reference

DSETTINGS
Sets grid and snap, polar and object snap tracking, object snap modes, and Dynamic Input.

FILL
Controls the filling of objects such as hatches, 2D solids, and wide polylines.

LWEIGHT
Sets the current lineweight, lineweight display options, and lineweight units.

QTEXT
Controls the display and plotting of text and attribute objects.

REGEN
Regenerates the entire drawing from the current viewport.

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

LWDISPLAY
Controls whether the lineweights of objects are displayed.

QTEXTMODE
Controls how text is displayed.

TEXTFILL
Controls the filling of TrueType fonts while printing.

TEXTQLTY
Sets the resolution tessellation fineness of text outlines.

Control the Transparency of Objects

You can control the transparency level of objects and layers.

Set the transparency level of selected objects or layers to enhance drawings or reduce the visibility of areas that are included for reference only.
Transparency can be set to ByLayer, ByBlock, or to a specific value.

IMPORTANT For performance reasons, plotting transparency is disabled by default. To plot transparent objects, check the Plot Transparency option in either the Print dialog box or Page Setup dialog box.

**Quick Reference**

CHPROP
Changes the properties of an object.

CHANGE
Changes the properties of existing objects.

LAYER
Manages layers and layer properties.

CETRANSPARENCY
Sets the transparency level for new objects.
TRANSPARENCYDISPLAY
Controls whether the object transparency is displayed.

Control How Overlapping Objects Are Displayed
You can control which overlapping objects appear to be on top.

Generally, overlapping objects such as text, wide polylines, and solid-filled polygons are displayed in the order they are created: newly created objects in front of existing objects.

You can use DRAWORDER to change the draw order (which is the display and plotting order) of any objects. TEXTTOFRONT changes the draw order of all text and dimensions in the drawing. HATCHTOBACK changes the draw order of hatches and gradient and solid fills in the drawing.

NOTE Overlapping objects cannot be controlled between model space and paper space. They can be controlled only within the same space.

Quick Reference

DRAWORDER
Changes the draw order of images and other objects.

HATCHTOBACK
Sets the draw order for all hatches in the drawing to be behind all other objects.

QSELECT
Creates a selection set based on filtering criteria.

REGEN
Regenerates the entire drawing from the current viewport.
SELECT
Places selected objects in the Previous selection set.

TEXTTOFRONT
Brings text and dimensions in front of all other objects in the drawing.

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

DRAWORDERCTL
Controls the default display behavior of overlapping objects when they are created or edited.

HPDRAWORDER
Controls the draw order of hatches and fills.

SORTENTS
Controls object sorting in support of draw order for several operations.

Control the Display of Objects
Control the display of objects by isolating or hiding a selection set.

Use ISOLATEOBJECTS and HIDEOBJECTS to create a temporary drawing view with selected objects isolated or hidden. This saves you the time of having to track objects across layers. If you isolate objects, only the isolated objects appear in the view. Use UNISOLATEOBJECTS to redisplay the objects. When you close and reopen the drawing, all previously hidden objects are displayed. Use the OBJECTISOLATIONMODE system variable to control whether objects remain hidden between drawing sessions.

Quick Reference

HIDEOBJECTS
Hides selected objects.

ISOLATEOBJECTS
Displays selected objects across layers; unselected objects are hidden.

UNISOLATEOBJECTS
Displays previously hidden objects.
OBJECTISOLATIONMODE

Controls whether hidden objects remain hidden between drawing sessions.
Use Precision Tools

You can use a variety of precision drawing tools to help you produce accurate drawings quickly and without performing tedious calculations.

Use Coordinates and Coordinate Systems (UCS)

For precise coordinate input, you can use several coordinate system entry methods. You can also use a movable coordinate system, the user coordinate system (UCS), for convenient coordinate entry and to establish workplanes.

Overview of Coordinate Entry

When a command prompts you for a point, you can use the pointing device to specify a point, or you can enter a coordinate value at the Command prompt. When dynamic input is on, you can enter coordinate values in tooltips near the cursor. You can enter two-dimensional coordinates as either Cartesian (X,Y) or polar coordinates.

Cartesian and Polar Coordinates

A Cartesian coordinate system has three axes, X, Y, and Z. When you enter coordinate values, you indicate a point's distance (in units) and its direction (+ or -) along the X, Y, and Z axes relative to the coordinate system origin (0,0,0).

In 2D, you specify points on the XY plane, also called the workplane. The workplane is similar to a flat sheet of grid paper. The X value of a Cartesian coordinate specifies horizontal distance, and the Y value specifies vertical distance. The origin point (0,0) indicates where the two axes intersect.

Polar coordinates use a distance and an angle to locate a point. With both Cartesian and polar coordinates, you can enter absolute coordinates based on the origin (0,0), or relative coordinates based on the last point specified.
Another method of entering a relative coordinate is by moving the cursor to specify a direction and then entering a distance directly. This method is called direct distance entry.

You can enter coordinates in scientific, decimal, engineering, architectural, or fractional notation. You can enter angles in grads, radians, surveyor’s units, or degrees, minutes, and seconds. The UNITS command controls unit format.

See also:

- Enter Cartesian Coordinates
- Enter Polar Coordinates
- Enter 3D Coordinates
- Use Dynamic Input on page 223

Quick Reference

BLIPMODE
  Controls the display of marker blips.
ID
  Displays the UCS coordinate values of a specified location.
LIST
  Displays property data for selected objects.
BLIPMODE
  Controls the display of marker blips.
LASTPOINT
  Stores the last point specified, expressed as UCS coordinates for the current space.

Enter 2D Coordinates

Absolute and relative 2D Cartesian and polar coordinates determine precise locations of objects in a drawing.
**Enter Cartesian Coordinates**

You can use absolute or relative Cartesian (rectangular) coordinates to locate points when creating objects.

To use Cartesian coordinates to specify a point, enter an $X$ value and a $Y$ value separated by a comma $(X,Y)$. The $X$ value is the positive or negative distance, in units, along the horizontal axis. The $Y$ value is the positive or negative distance, in units, along the vertical axis.

Absolute coordinates are based on the UCS origin $(0,0)$, which is the intersection of the $X$ and $Y$ axes. Use absolute coordinates when you know the precise $X$ and $Y$ values of the point.

With dynamic input, you can specify absolute coordinates with the `#` prefix. If you enter coordinates on the command line instead of in the tooltip, the `#` prefix is not used. For example, entering `#3,4` specifies a point 3 units along the $X$ axis and 4 units along the $Y$ axis from the UCS origin. For more information about dynamic input, see Use Dynamic Input on page 223.

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:

![Graph](image)

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

  #x,y

  If dynamic input is turned off, enter coordinates on the command line using the following format:

  X Y

  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 UNITS.

Absolute polar coordinates are measured from the UCS origin (0,0), which is the intersection of the X and Y axes. Use absolute polar coordinates when you know the precise distance and angle coordinates of the point.

With dynamic input, you can specify absolute coordinates with the # prefix. If you enter coordinates on the command line instead of in the tooltip, the # prefix is not used. For example, entering #3<45 specifies a point 3 units from the origin at an angle of 45 degrees from the X axis. For more information about dynamic input, see Use Dynamic Input on page 223.

The following example shows two lines drawn with absolute polar coordinates using the default angle direction setting. Enter the following in the tooltip:

Command: line
From point: #0,0
To point: #4<120
To point: #5<30
Relative coordinates are based on the last point entered. Use relative coordinates when you know the location of a point in relation to the previous point.

To specify relative coordinates, precede the coordinate values with an @ sign. For example, entering @1<45 specifies a point at a distance of 1 unit from the last point specified at an angle of 45 degrees from the X axis.

The following example shows two lines drawn with relative polar coordinates. In each illustration, the line begins at the location labeled as the previous point.

Command: line
From point: @3<45
To point: @5<285

To enter absolute polar coordinates (2D)
- At a prompt for a point, enter coordinates in the tooltip using the following format:
  #distance<angle
  
  If dynamic input is turned off, enter coordinates on the command line using the following format:
  distance<angle

To enter relative polar coordinates (2D)
- At a prompt for a point, enter coordinates using the following format:
  @distance<angle
Quick Reference

UNITs
Controls coordinate and angle display formats and precision.

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.

![3D Cartesian Coordinate Illustration](image)
Use Default Z Values

When you enter coordinates in the format X,Y, the Z value is copied from the last point you entered. As a result, you can enter one location in the X,Y,Z format and then enter subsequent locations using the X,Y format with the Z value remaining constant. For example, if you enter the following coordinates for a line

From point: **0,0,5**
To point: **3,4**

both endpoints of the line will have a Z value of 5. When you begin or open any drawing, the initial default value of Z is greater than 0.

Use Absolute and Relative Coordinates

As with 2D coordinates, you can enter absolute coordinate values, which are based on the origin, or you can enter relative coordinate values, which are based on the last point entered. To enter relative coordinates, use the @ sign as a prefix. For example, use @1,0,0 to enter a point one unit in the positive X direction from the previous point. To enter absolute coordinates at the Command prompt, no prefix is necessary.

**To enter absolute coordinates (3D)**

- At a prompt for a point, enter coordinates in the tooltip using the following format:
  
  \#x,y,z
  
  If dynamic input is turned off, enter coordinates on the command line using the following format:
  
  x,y,z

**To enter relative coordinates (3D)**

- At a prompt for a point, enter coordinates using the following format:
  
  @x,y,z

Quick Reference

GRID

Displays a grid pattern in the current viewport.
SNAP
Restricts cursor movement to specified intervals.

UCS
Manages user coordinate systems.

UCSICON
Controls the visibility and placement of the UCS icon.

UNITS
Controls coordinate and angle display formats and precision.

ELEVATION
Stores the current elevation of new objects relative to the current UCS.

Enter Cylindrical Coordinates
3D cylindrical coordinates describe a precise location by a distance from the UCS origin in the XY plane, an angle from the X axis in the XY plane, and a Z value.

Cylindrical coordinate entry is the 3D equivalent of 2D polar coordinate entry. It specifies an additional coordinate on an axis that is perpendicular to the XY plane. Cylindrical coordinates define points by a distance in the XY plane from the UCS origin, an angle from the X axis in the XY plane, and a Z value. You specify a point using absolute cylindrical coordinates with the following syntax:

\[ \text{X-\{angle from X axis\}, Z} \]

**NOTE** For the following examples, it is assumed that dynamic input is turned off or that the coordinates are entered on the command line. With dynamic input, you specify absolute coordinates with the `#` prefix.

In the illustration below, 5<30,6 indicates a point 5 units from the origin of the current UCS, 30 degrees from the X axis in the XY plane, and 6 units along the Z axis.
When you need to define a point based on a previous point rather than the UCS origin, you can enter relative cylindrical coordinate values with the @ prefix. For example, @4<45,5 specifies a point 4 units in the XY plane from the last point entered, at an angle of 45 degrees from the positive X direction, and extending 5 units in the positive Z direction.

**To enter relative cylindrical coordinates**

- At a prompt for a point, enter the coordinate values using the following format:
  
  @x<angle from the X axis,z

For example, @4<60,2 represents a location that is 4 units along the X axis from the last point measured at 60 degrees from the positive X axis and at 2 units in the positive Z direction.

**Quick Reference**

**UNITS**

Controls coordinate and angle display formats and precision.

**Enter Spherical Coordinates**

3D spherical coordinates specify a location by a distance from the origin of the current UCS, an angle from the X axis in the XY plane, and an angle from the XY plane.
Spherical coordinate entry in 3D is similar to polar coordinate entry in 2D. You locate a point by specifying its distance from the origin of the current UCS, its angle from the $X$ axis (in the $XY$ plane), and its angle from the $XY$ plane, each angle preceded by an open angle bracket (<) as in the following format:

$X<$[angle from $X$ axis]<[angle from $XY$ plane]

**NOTE** For the following examples, it is assumed that dynamic input is turned off or that the coordinates are entered on the command line. With dynamic input, you specify absolute coordinates with the # prefix.

In the following illustration, $8<60<30$ indicates a point 8 units from the origin of the current UCS in the $XY$ plane, 60 degrees from the $X$ axis in the $XY$ plane, and 30 degrees up the $Z$ axis from the $XY$ plane. $5<45<15$ indicates a point 5 units from the origin, 45 degrees from the $X$ axis in the $XY$ plane, and 15 degrees up from the $XY$ plane.

When you need to define a point based on a previous point, enter the relative spherical coordinate values by preceding them with the @ sign.

**To enter relative spherical coordinates**

- At a prompt for a point, enter the coordinate values using the following format:
  
  $@x<$angle from the $x$ axis$<$angle from the $xy$ plane

For example, $@4<60<30$ represents a location that is 4 units from the last point measured at 60 degrees from the positive $X$ axis in the $XY$ plane and at 30 degrees from the $XY$ plane.
Quick Reference

UNITS
Controls coordinate and angle display formats and precision.

Understand the User Coordinate System (UCS)
You can relocate and rotate the user coordinate system for convenient coordinate entry, grid display, grid snap, Ortho mode, and other drawing tools.

Understand the World and User Coordinate Systems
There are two coordinate systems: a fixed system called the world coordinate system (WCS) and a movable system called the user coordinate system (UCS). By default, these two systems are coincident in a new drawing.

Normally in 2D views, the WCS X axis is horizontal and the Y axis is vertical. The WCS origin is where the X and Y axes intersect (0,0). All objects in a drawing file are defined by their WCS coordinates. However, it is usually more convenient to create and edit objects based on the movable UCS.

Work with the User Coordinate System
Virtually all coordinate entry as well as many other tools and operations reference the current UCS. 2D tools and operations that depend on the location and orientation of the UCS include the following:

■ Absolute and relative coordinate entry
■ Absolute reference angles
■ Definition of horizontal and vertical for Ortho mode, polar tracking, object snap tracking, grid display, and grid snap
■ Orientation of horizontal and vertical dimensions
■ Orientation of text objects
■ View rotation using the PLAN command

Moving or rotating the UCS can make it easier to work on particular areas of a drawing.
You can relocate the user coordinate system with methods such as the following:

- Move the UCS by defining a new origin point.
- Align the UCS with an existing object.
- Rotate the UCS by specifying a new origin point and a point on the new X axis.
- Rotate the current UCS a specified angle around the Z axis.
- Revert to the previous UCS.
- Restore the UCS to be coincident with the WCS.

Each of these methods have a corresponding option in the UCS command. Once you have defined a UCS, you can name it and then restore it when you need to use it again.

**To define a new UCS origin in 2D**

1. Click View tab ➤ Coordinates panel ➤ Origin.
2. Specify a point for the new origin. The UCS origin (0,0) is redefined at the point you specify.

**To change the rotation angle of the UCS**

1. Click View tab ➤ Coordinates panel ➤ Z.
2. Specify a rotation angle.

**To restore the UCS to be coincident with the WCS**

1. Click View tab ➤ Coordinates panel ➤ Named UCS.
2. In the UCS dialog box, Named UCSs tab, select World.
3. Click Set Current.
4. Click OK.
To restore the previous UCS

1. Click View tab ➤ Coordinates panel ➤ Named UCS.
2. In the UCS dialog box, Named UCSs tab, select Previous.
3. Click Set Current.
4. Click OK.

To save a UCS

1. Click View tab ➤ Coordinates panel ➤ Named UCS.
   The new UCS is displayed in the UCS list as UNNAMED.
2. In the UCS dialog box, Named UCSs tab, select UNNAMED and enter a new name. (You can also select UNNAMED, and right-click. Click Rename.)
3. Click OK.
   You can use up to 255 characters, including letters, digits, and the special characters dollar sign ($), hyphen (-), and underscore (_). All UCS names are converted to uppercase.

To restore a named UCS

1. Click View tab ➤ Coordinates panel ➤ Named UCS.
2. In the UCS dialog box, Named UCSs tab, you can view the origin and axis direction of a listed UCS. Select the UCS name. Click Details.
   When you have finished viewing the list, click OK to return to the UCS dialog box.
3. Select the coordinate system you want to restore. Click Set Current.
4. Click OK.

To rename a UCS

1. Click View tab ➤ Coordinates panel ➤ Named UCS.
2. In the UCS dialog box, Named UCSs tab, select the coordinate system you want to rename. (You can also select UNNAMED, and right-click. Click Rename.)
3. Enter a new name.
4  Click OK.

To delete a named UCS

1  Click View tab ➤ Coordinates panel ➤ Named UCS.
2  In the UCS dialog box, Named UCSs tab, select the UCS you want to delete.
3  Press Delete.
   You cannot delete the current UCS or a UCS with the default name UNNAMED.

Quick Reference

UCS
Manages user coordinate systems.

UCSICON
Controls the visibility and placement of the UCS icon.

UCSMAN
Manages defined user coordinate systems.

ERRNO
Displays the number of the appropriate error code when an AutoLISP function call causes an error that AutoCAD detects.

PUCSBASE
Stores the name of the UCS that defines the origin and orientation of orthographic UCS settings in paper space only.

UCSFOLLOW
Generates a plan view whenever you change from one UCS to another.

UCSNAME
Stores the name of the current coordinate system for the current viewport in the current space.

UCSORG
Stores the origin point of the current coordinate system for the current viewport in the current space.
UCSORTHO
Determined whether the related orthographic UCS setting is restored automatically when an orthographic view is restored.

UCSXDIR
Stores the X direction of the current UCS for the current viewport in the current space.

UCSYDIR
Stores the Y direction of the current UCS for the current viewport in the current space.

Specify Workplanes in 3D (UCS)
Control of the user coordinate system is essential for effective 3D modeling.

Understand the User Coordinate System in 3D
When you work in 3D, the user coordinate system is useful for entering coordinates, creating 3D objects on 2D workplanes, and rotating objects in 3D.

When you create or modify objects in a 3D environment, you can move and reorient the UCS in 3D model space to simplify your work. The $XY$ plane of the UCS is called the workplane.

Important operations on objects in a 3D environment that depend on the location and orientation of the UCS include the following:

- Establish the workplane in on which to create and modify objects
- Establish the workplane that contains the grid display and grid snap
- Establish a new UCS $Z$ axis about which to rotate objects in 3D
- Determine up and down directions as well as horizontal and vertical for Ortho mode, polar tracking, and object snap tracking
- Define a 3D view directly into the workplane with the PLAN command

Apply the Right-Hand Rule
Use the right-hand rule to determine the positive axis direction of the $Z$ axis when you know the direction of the $X$ and $Y$ axes in a 3D coordinate system.
Place the back of your right hand near the screen and point your thumb in the direction of the positive X axis. Extend your index and middle fingers as illustrated, pointing your index finger in the direction of the positive Y axis. Your middle finger indicates the direction of the positive Z axis. By rotating your hand, you see how the X, Y, and Z axes rotate as you change the UCS.

You can also use the right-hand rule for determining the default positive direction of rotation about an axis in 3D space. Point your right thumb in the positive direction of the axis and curl your fingers. Your fingers indicate the positive rotation direction about the axis.

\[\begin{array}{c}
\text{Y} \\
\text{Z} \\
\text{X}
\end{array}\]

NOTE By default, when you specify a view in 3D, it is established relative to the fixed WCS rather than the movable UCS.

Quick Reference

UCS

Manages user coordinate systems.

UCSICON

Controls the visibility and placement of the UCS icon.

UCSMAN

Manages defined user coordinate systems.

UCSICON

Displays the UCS icon for the current viewport or layout.

UCSVP

Determines whether the UCS in viewports remains fixed or changes to reflect the UCS of the current viewport.
Control the User Coordinate System in 3D

Several methods are available for manipulating the user coordinate system in 3D. You can also save and restore user coordinate system orientations.

You define a user coordinate system (UCS) to change the location of the 0,0,0 origin point, the location and rotation of the XY plane, and the orientation of the XY plane or Z axis. You can locate and orient a UCS anywhere in 3D space, and you can define, save, and recall as many saved UCS locations as you require.

If multiple viewports are active, you can assign a different UCS to each viewport. With the UCSVP system variable turned on, you can lock a UCS to a viewport, automatically restoring the UCS each time that viewport is made current.

Define the UCS Location

You can define a UCS in the following ways:

- Specify a new origin (one point), new X axis (two points), or new XY plane (three points).
- Align the UCS by selecting a face on a 3D solid object. The selection can be on a face or on an edge of the solid.
- Align the new UCS with an existing object. The origin of the UCS is located at the vertex nearest to where the object was selected.
- Align the new UCS with the current viewing direction.
- Rotate the current UCS around any of its three major axes.
- Reorient the XY plane by specifying a new Z axis.

Use UCS Presets

If you do not want to define your own UCS, you can choose from several preset coordinate systems. The images on the Orthographic tab of the UCS dialog box show the available choices.

Change the Default Elevation

The ELEV command sets the default Z value for new objects above or below the XY plane of the current UCS. This value is stored in the ELEVATION system variable.
NOTE  Generally, it is recommended that you leave the elevation set to zero and control the XY plane of the current UCS with the UCS command.

Change the UCS in Paper Space

You can define a new UCS in paper space just as you can in model space; however, the UCS in paper space is restricted to 2D manipulation. Although you can enter 3D coordinates in paper space, you cannot use 3D viewing commands such as PLAN and VPOINT.

Save and Restore UCS Locations by Name

If you plan to work extensively in 3D, you can save named UCS locations, each having a different origin and orientation, for various construction requirements. You can relocate, save, and recall as many UCS orientations as you require.

Quick Reference

ELEV
    Sets elevation and extrusion thickness of new objects.

UCS
    Manages user coordinate systems.

UCSMAN
    Manages defined user coordinate systems.

ELEVATION
    Stores the current elevation of new objects relative to the current UCS.

UCSVP
    Determines whether the UCS in viewports remains fixed or changes to reflect the UCS of the current viewport.

Use the Dynamic UCS with Solid Models

With the dynamic UCS feature, you can temporarily and automatically align the XY plane of the UCS with a plane on a solid model while creating objects.
When in a draw command, you align the UCS by moving your pointer over an edge of a face rather than having to use the UCS command. After you finish the command, the UCS returns to its previous location and orientation.

For example, you can use the dynamic UCS to create a rectangle on an angled face of a solid model as shown in the illustration.

In the illustration on the left, the UCS is not aligned with the angled face. Instead of relocating the UCS, you turn on the dynamic UCS on the status bar or by pressing Fn-F6.

When you move the pointer completely over an edge as shown in the middle illustration, the cursor changes to show the direction of the dynamic UCS axes. You can then create objects on the angled face easily as shown in the illustration on the right.

**NOTE** To display the XYZ labels on the cursor, right-click the DUCS button and click Display Crosshair Labels.

The X axis of the dynamic UCS is located along an edge of the face and the positive direction of the X axis always points toward the right half of the screen. Only the front faces of a solid are detected by the dynamic UCS.

The types of commands that can use a dynamic UCS include the following:

- **Simple geometry.** Line, polyline, rectangle, arc, circle
- **Text.** Text, Multiline text, table
- **References.** Insert, xref
Solids. Primitives and POLYSOLID

Editing. Rotate, mirror, align

Other. UCS, area, grip tool manipulation

TIP You can easily align the UCS with a plane on a solid model by turning on the dynamic UCS feature and then using the UCS command to locate the origin on that plane.

If Grid and Snap mode are turned on, they align temporarily to the dynamic UCS. The limits of the grid display are set automatically.

You can temporarily turn off the dynamic UCS by pressing Fn-F6 or Shift-Z while moving the pointer over a face.

NOTE The dynamic UCS is available only while a command is active.

Quick Reference

OPTIONS
Customizes the program settings.

OSOPTIONS
Automatically suppresses object snaps on hatch objects and geometry with negative Z values when using a dynamic UCS.

UCSDETECT
Controls whether dynamic UCS acquisition is active or not.

Assign User Coordinate System Orientations to Viewports

To facilitate editing objects in different views, you can define a different user coordinate system orientation for each view.

Multiple viewports provide different views of your model. For example, you might set up viewports that display top, front, right side, and isometric views. To facilitate editing objects in different views, you can define a different UCS for each view. Each time you make a viewport current, you can begin drawing using the same UCS you used the last time that viewport was current.

The UCS in each viewport is controlled by the UCSVP system variable. When UCSVP is set to 1 in a viewport, the UCS last used in that viewport is saved.
with the viewport and is restored when the viewport is made current again. When UCSVP is set to 0 in a viewport, its UCS is always the same as the UCS in the current viewport.

For example, you might set up three viewports: a top view, front view, and isometric view. If you set the UCSVP system variable to 0 in the isometric viewport, you can use the Top UCS in both the top viewport and the isometric viewport. When you make the top viewport current, the isometric viewport's UCS reflects the UCS top viewport. Likewise, making the front viewport current switches the isometric viewport's UCS to match that of the front viewport.

The example is illustrated in the following figures. The first figure shows the isometric viewport reflecting the UCS of the upper-left, or top, viewport, which is current.

![First Figure](image1.png)

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.

![Second Figure](image2.png)
In previous releases, the UCS was a global setting for all viewports in either model or paper space. If you want to restore the behavior of earlier releases, you can set the value of the UCSVP system variable to 0 in all active viewports.

**Quick Reference**

**UCS**
- Manages user coordinate systems.

**UCSVP**
- Determines whether the UCS in viewports remains fixed or changes to reflect the UCS of the current viewport.

**Control the Display of the User Coordinate System Icon**

To help visualize the current orientation of the user coordinate system, you can display the user coordinate system icon. Several versions of this icon are available, and you can change its size, location, and color.

To indicate the location and orientation of the UCS, the UCS icon is displayed either at the UCS origin point or in the lower-left corner of the current viewport.

You can choose a 2D or 3D style of the icon to represent the UCS when working in 2D environment. Shaded style of icon is displayed to represent the UCS in the 3D environment.

![UCS Icon Variations](image)

Use the UCSICON command to choose between displaying the 2D or the 3D UCS icon. The shaded UCS icon is displayed for a shaded 3D view. To indicate the origin and orientation of the UCS, you can display the UCS icon at the UCS origin point using the UCSICON command.

If you have multiple viewports, each viewport displays its own UCS icon. The UCS icon is displayed in various ways to help you visualize the orientation of the workplane. The following figure shows some of the possible icon displays.
You can use the UCSICON command to switch between the 2D UCS icon and the 3D UCS icon. You can also use the command to change the size, color, and icon line width of the 3D UCS icon.

The UCS broken pencil icon replaces the 2D UCS icon when the viewing direction is in a plane parallel to the UCS XY plane. The broken pencil icon indicates that the edge of the XY plane is almost perpendicular to your viewing direction. This icon warns you not to use your pointing device to specify coordinates.

When you use the pointing device to locate a point, it’s normally placed on the XY plane. If the UCS is rotated so that the Z axis lies in a plane parallel to the viewing plane—that is, if the XY plane is edge-on to the viewer—it may be difficult to visualize where the point will be located. In this case, the point will be located on a plane parallel to your viewing plane that also contains the UCS origin point. For example, if the viewing direction is along the X axis, coordinates specified with a pointing device will be located on the YZ plane, which contains the UCS origin point.

Use the 3D UCS icon to help you visualize which plane these coordinates will be projected on; the 3D UCS icon does not use a broken pencil icon.

**Quick Reference**

UCSICON

Controls the visibility and placement of the UCS icon.

UCSICON

Displays the UCS icon for the current viewport or layout.
Use Dynamic Input

Dynamic Input provides a command interface near the cursor to help you keep your focus in the drafting area.

When dynamic input is on, tooltips display information near the cursor that is dynamically updated as the cursor moves. When a command is active, the tooltips provide a place for user entry.

After you type a value in an input field and press Tab, the field then displays a lock icon, and the cursor is constrained by the value that you entered. You can then enter a value for the second input field. Alternately, if you type a value and press Enter, the second input field is ignored and the value is interpreted as direct distance entry.

The actions required to complete a command or to use grips are similar to those for the Command prompt. The difference is that your attention can stay near the cursor.

Dynamic input is not designed to replace the command line. You can hide the command line to add screen area for drawing, but you will need to display it for some operations.

Turn On or Turn Off Dynamic Input

Click the dynamic input button on the status bar to turn dynamic input on and off. Dynamic input has three components: pointer input, dimensional input, and dynamic prompts. Right-click the dynamic input button and click Settings to control what is displayed by each component when dynamic input is on.

Pointer Input

When pointer input is on and a command is active, the location of the crosshairs is displayed as coordinates in a tooltip near the cursor. You can enter coordinate values in the tooltip instead of on the command line.

The default for second and subsequent points is relative polar coordinates (relative Cartesian for RECTANG). There is no need to type the at sign (@). If you want to use absolute coordinates, use the pound sign (#) prefix. For example, to move an object to the origin, for the second point prompt, enter #0,0.
Use the pointer input settings to change the default format for coordinates and to control when pointer input tooltips are displayed.

**Dimensional Input**

When dimensional input is on, the tooltips display distance and angle values when a Command prompts for a second point. The values in the dimensional tooltips change as you move the cursor. Press Tab to move to the value you want to change. Dimensional input is available for ARC, CIRCLE, ELLIPSE, LINE, and PLINE.

When you use grips to edit an object, the dimensional input tooltips can display the following information:

- Original length
- A length that updates as you move the grip
- The change in the length
- Angle
- The change in the angle as you move the grip
- The radius of an arc
Use the dimensional input settings to display only the information you want to see.

When you use grips to stretch objects or when you create new objects, dimensional input displays only acute angles, that is, all angles are displayed as 180 degrees or less. Thus, an angle of 270 degrees is displayed as 90 degrees regardless of the ANGDIR system variable setting (set in the Drawing Units dialog box). Angles specified when creating new objects rely on the cursor location to determine the positive angle direction.

**Dynamic Prompts**

When dynamic prompts are on, prompts are displayed in a tooltip near the cursor. You can enter a response in the tooltip instead of on the command line. Press the Down Arrow key to view and select options. Press the Up Arrow key to display recent input.

**NOTE** To use paste text into a dynamic prompt tooltip, type a letter and then backspace to delete it before you paste the entry. Otherwise, the entry is pasted into the drawing as text.

**Quick Reference**

**DSETTINGS**

Sets grid and snap, polar and object snap tracking, object snap modes, and Dynamic Input.

**DYNDIGRIP**

Controls which dynamic dimensions are displayed during grip stretch editing.

**DYNDIVIS**

Controls how many dynamic dimensions are displayed during grip stretch editing.
DYNMODE
Turns Dynamic Input features on and off.

DYNPICOORDS
Controls whether pointer input uses relative or absolute format for coordinates.

DYNPIFORMAT
Controls whether pointer input uses polar or Cartesian format for coordinates.

DYNPIVIS
Controls when pointer input is displayed.

DYNPROMPT
Controls display of prompts in Dynamic Input tooltips.

DYNTOOLTIPS
Controls which tooltips are affected by tooltip appearance settings.

TEMPOVERRIDES
Turns temporary override keys on and off.

TOOLTIPMERGE
Combines drafting tooltips into a single tooltip.

Snap to Locations on Objects (Object Snaps)
Instead of entering coordinates, you can specify points relative to existing objects such as endpoints of lines or center points of circles.

Use Object Snaps
Use object snaps to specify precise locations on objects. For example, you can use an object snap to draw a line to the center of a circle or to the midpoint of a polyline segment.

You can specify an object snap whenever you are prompted for a point. By default, a marker and a tooltip are displayed when you move the cursor over an object snap location on an object. This feature, called AutoSnap™, provides a visual clue that indicates which object snaps are in effect.
For a list of object snaps, see OSNAP.

**Specify an Object Snap**

To specify an object snap at a prompt for a point, you can

- When prompted for a point, right-click and choose an object snap from the Snap Overrides sub-menu
- Enter the name of an object snap at the Command prompt
- On the status bar, right-click the object snap button

When you specify an object snap at a prompt for a point, the object snap stays in effect only for the next point that you specify.

**NOTE** Object snaps work only when you are prompted for a point. If you try to use an object snap at the Command prompt, an error message is displayed.

**Use Running Object Snaps**

If you need to use one or more object snaps repeatedly, you can turn on *running object snaps*. For example, you might set Center as a running object snap if you need to connect the centers of a series of circles with a line.

You can specify one or more running object snaps on the Object Snaps tab in the Drafting Settings dialog box, which is accessible from the Tools menu. If several running object snaps are on, more than one object snap may be eligible at a given location. Press Tab to cycle through the possibilities before you specify the point.

Click the object snap button on the status bar or press Fn-F3 to turn running object snaps on and off.

**NOTE** If you want object snaps to ignore hatch objects, set the OSOPTIONS system variable to 1.
Use Object Snaps in 3D

By default, the Z-value of an object snap location is determined by the object’s location in space. However, if you work with object snaps on the plan view of a building or the top view of a part, a constant Z-value is more useful.

If you turn on the OSNAPZ system variable, all object snaps are projected onto the XY plane of the current UCS or, if ELEV is set to a non-zero value, onto a plane parallel to XY plane at the specified elevation.

NOTE When you draw or modify objects, make sure that you know whether OSNAPZ is on or off. There is no visual reminder, and you can get unexpected results.

Quick Reference

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

OPTIONS
- Customizes the program settings.

OSNAP
- Sets running object snap modes.

APBOX
- Turns the display of the AutoSnap aperture box on or off.

AUTOSNAP
- Controls the display of the AutoSnap marker, tooltip, and magnet.

OSMODE
- Sets running object snaps

OSNAPZ
- Controls whether object snaps are automatically projected onto a plane parallel to the XY plane of the current UCS at the current elevation.

OSNAPCOORD
- Controls whether coordinates entered on the command line will override running object snaps.
OSOPTIONS
Automatically suppresses object snaps on hatch objects and geometry with negative Z values when using a dynamic UCS.

MTP (Command Modifier)
Locates the midpoint between two points.

The Object Snap Menu
Specify an object snap quickly and conveniently from a shortcut menu.
The object snap menu is displayed at your cursor location when you right-click while being prompted for a point and click Snap Overrides.

See also:
- Use Object Snaps on page 226

Quick Reference
OSNAP
Sets running object snap modes.

Set Visual Aids for Object Snaps (AutoSnap)
Object snaps include a visual aid called AutoSnap™ to help you see and use object snaps more efficiently. AutoSnap displays a marker and a tooltip when you move your cursor over an object snap location.

AutoSnap Tools
AutoSnap consists of the following snap tools:

- **Marker.** Displays the object snap location when the cursor moves over or near an object. Marker shape is dependent on the snap it is marking.

- **Tooltip.** Describes which part of the object you are snapping to in a small flag at the cursor location.

- **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 and tooltips are turned on by default. You can change AutoSnap marker size on the Cursor & Selection tab in the Application Preferences 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.

**Quick Reference**

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

**OPTIONS**
Customizes the program settings.

**OSNAP**
Sets running object snap modes.

**APBOX**
Turns the display of the AutoSnap aperture box on or off.

**AUTOSNAP**
Controls the display of the AutoSnap marker, tooltip, and magnet.

**OSMODE**
Sets running object snaps

**Override Object Snap Settings**

While you work, you can turn running object snaps on and off temporarily by using an override key. Temporary override keys can also be used for other drawing aids; for example, Ortho mode and Polar mode.

For example, if you have set running object snaps but you want to turn them off for one point, you can hold down Fm-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:

<p>| | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Object snap override: Endpoint</td>
<td>5</td>
<td>Turns off all snapping and tracking</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>Enforces object snap selection</td>
<td>6</td>
<td>Object snap override: Center</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>Toggles object snap tracking</td>
<td>7</td>
<td>Object snap override: Midpoint</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>Toggles object snap mode (OSNAP)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

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”

Quick Reference

CUI
Manages the customized user interface elements in the product.

OPTIONS
Customizes the program settings.

OSNAP
Sets running object snap modes.

OSMODE
Sets running object snaps

TEMPOVERRIDES
Turns temporary override keys on and off.

Restrict Cursor Movement
Several tools are available that you can use to restrict or lock the movement of your cursor.

Adjust Grid and Grid Snap
To enhance drawing speed and efficiency, you can display and snap to a rectangular grid. You can also control its spacing, angle, and alignment.

The grid is a rectangular pattern of dots or lines that extends over the area you specify as the grid limits. Using the grid is similar to placing a sheet of grid paper under a drawing. The grid helps you align objects and visualize the distances between them. The grid is not plotted.

Snap mode restricts the movement of the crosshairs to intervals that you define. When Snap mode is on, the cursor seems to adhere, or “snap,” to an
invisible rectangular grid. Snap is useful for specifying precise points with the arrow keys or the pointing device.

Grid mode and Snap mode are independent but are often turned on at the same time.

**Control the Display Style and Area of the Grid**

You can display the grid either as a rectangular pattern of dots or as rectangular pattern of lines. The grid displays lines for all visual styles. The grid displays dots only when the current visual style is set to 2D Wireframe. By default, a lined grid is displayed while working in both 2D and 3D. There are several methods to change the current visual style, including the `VSCURRENT` command.

The `LIMITS` command controls the drawing area covered by the grid. As an option, you can override the limits to make the grid cover the entire $XY$ plane of the user coordinate system (UCS). You can access this option in the Drafting Settings dialog box or use the `GRIDDISPLAY` system variable.

---

**NOTE** When you use dynamic UCS, the grid limits are set automatically relative to the size of the selected face of the solid and the drawing area available.

**Control the Frequency of Major Grid Lines**

If the grid is displayed as lines rather than dots, darker lines called *major grid lines* display at intervals. When working in decimal units or with feet and inches, major grid lines are especially useful for measuring distances quickly. You can control the frequency of major grid lines in the Drafting Settings.
To turn off the display of major grid lines, set the frequency of major grid lines to 1.

**NOTE** If the grid is displayed as lines, the grid limits are displayed also as darker lines. Do not confuse these boundaries with major grid lines.

**NOTE** When the grid is displayed as lines and SNAPANG is set to a value other than 0, the grid will not display. SNAPANG does not affect the display of the dotted grid.

**Change the Grid Dynamically During Zooming**

If you zoom in or out of your drawing, the grid spacing is adjusted automatically to be more appropriate for the new magnification. This is called *adaptive grid display*.

For example, if you zoom way out, the density of displayed grid lines reduces automatically. Conversely, if you zoom way in, additional grid lines display in the same proportion as the major grid lines.
Change Grid and Snap Spacing

As you work, you can turn Grid and Snap mode on and off, and you can change the grid and snap spacing. You can turn Snap mode on and off temporarily by using an override key.

Snap spacing does not have to match grid spacing. For example, you might set a wide grid spacing to be used as a reference but maintain a closer snap spacing for accuracy in specifying points.

Change the Grid and Snap Angle and Base

If you need to draw along a specific alignment or angle, you can change the grid and snap angle by rotating the user coordinate system (UCS). This rotation realigns the crosshairs on the screen to match the new angle. In the following example, the UCS is rotated 30 degrees to match the angle of the anchor bracket.

The grid and snap points are always aligned with the UCS origin. If you need to shift the grid and grid snap origin, move the UCS.

See also:

- “Set Isometric Grid and Snap”
- Override Object Snap Settings on page 230
- Use a Visual Style to Display Your Model on page 92

Quick Reference

DSETTINGS

Sets grid and snap, polar and object snap tracking, object snap modes, and Dynamic Input.
GRID
Displays a grid pattern in the current viewport.

LIMITS
Sets and controls the limits of the grid display in the current Model or named layout.

SHADEMODE
Starts the VSCURRENT command.

SNAP
Restricts cursor movement to specified intervals.

GRIDDISPLAY
Controls the display behavior and display limits of the grid.

GRIDMODE
Specifies whether the grid is turned on or off.

GRIDMAJOR
Controls the frequency of major grid lines compared to minor grid lines.

GRIDUNIT
Specifies the grid spacing (X and Y) for the current viewport.

LIMCHECK
Controls the creation of objects outside the grid limits.

LIMMAX
Stores the upper-right grid limits for the current space, expressed as world coordinates.

LIMMIN
Stores the lower-left grid limits for the current space, expressed as a world coordinate.

SNAPANG
Sets the snap and grid rotation angle for the current viewport relative to the current UCS.
SNAPBASE
Sets the snap and grid origin point for the current viewport relative to the current UCS.

SNAPMODE
 Turns the Snap mode on and off.

SNAPTYPE
 Sets the type of snap for the current viewport.

SNAPUNIT
 Sets the snap spacing for the current viewport.

TEMPOVERRIDES
 Turns temporary override keys on and off.

Use Orthogonal Locking (Ortho Mode)
You can restrict cursor movement to horizontal and vertical for convenience and precision when creating and modifying objects.

As you create or move objects, you can use Ortho mode to restrict the cursor to the horizontal or vertical axis. As you move the cursor, the rubber-band line follows the horizontal or vertical axis, whichever is nearest the cursor.

The orientation of the current user coordinate system (UCS) determines the horizontal and vertical directions. In 3D views, Ortho mode additionally restricts the cursor to the up and down directions. In that case, the tooltip displays a +Z or -Z for the angle.

**TIP** Use direct distance entry with Ortho mode turned on to create orthogonal lines of specified lengths or to move objects specified distances.

You can turn Ortho on and off at any time during drawing and editing. Ortho is ignored when you enter coordinates or specify an object snap. To turn Ortho on or off temporarily, hold down the temporary override key, Shift. While you use the temporary override key, the direct distance entry method is not available.

For drawing or editing objects at angles that are not parallel to the horizontal or vertical axis, see Use Polar Tracking and PolarSnap on page 238.

If turned on, the isometric snap setting takes priority over the UCS in determining horizontal and vertical directions.
NOTE Ortho mode and polar tracking cannot be on at the same time. Turning on Ortho turns off polar tracking.

See also:
- Override Object Snap Settings on page 230

Quick Reference

ORTHO
Constrains cursor movement to the horizontal or vertical direction.

ORTHOMODE
Constrains cursor movement to the perpendicular.

TEMPOVERRIDES
Turns temporary override keys on and off.

Use Polar Tracking and PolarSnap

Polar tracking restricts cursor movement to specified angles. PolarSnap restricts cursor movement to specified increments along a polar angle.

When you are creating or modifying objects, you can use polar tracking to display temporary alignment paths defined by the polar angles you specify. In 3D views, polar tracking additionally provides an alignment path in the up and down directions. In that case, the tooltip displays a +Z or -Z for the angle.

Polar angles are relative to the orientation of the current user coordinate system (UCS) and the setting for the base angle convention in a drawing. The angle base direction is set in the Drawing Units dialog box (UNITS).

Use PolarSnap™ to snap to specified distances along the alignment path. For example, in the following illustration you draw a two-unit line from point 1 to point 2, and then draw a two-unit line to point 3 at a 45-degree angle to the line. If you turn on the 45-degree polar angle increment, an alignment path and tooltip are displayed when your cursor crosses the 0 or 45-degree angle. The alignment path and tooltip disappear when you move the cursor away from the angle.
As you move your cursor, alignment paths and tooltips are displayed when you move the cursor near polar angles. The default angle measurement is 90 degrees. Use the alignment path and tooltip to draw your object. You can use polar tracking with Intersection and Apparent Intersection object snaps to find where a polar alignment path intersects another object.

**NOTE** Ortho mode and polar tracking cannot be on at the same time. Turning on polar tracking turns off Ortho mode. Similarly, PolarSnap and grid snap cannot be on at the same time. Turning on PolarSnap turns off grid snap.

### Specify Polar Angles (Polar Tracking)

You can use polar tracking to track along polar angle increments of 90, 60, 45, 30, 22.5, 18, 15, 10, and 5 degrees, or you can specify other angles. The following illustration shows the alignment paths displayed as you move your cursor 90 degrees with the polar angle increment set to 30 degrees.

The orientation of 0 depends on the angle you set in the Drawing Units dialog box. The direction of snap (clockwise or counterclockwise) depends on the units direction you specify when setting units of measurement.

You can turn polar tracking on and off temporarily by using an override key. The direct distance entry method is not available while you are using the temporary override key for polar tracking.

### Specify Polar Distances (PolarSnap)

PolarSnap restricts cursor movement to increments of a polar distance you specify. For example, if you specify a length of 4 units, the cursor snaps from...
the first point specified to lengths of 0, 4, 8, 12, 16, and so on. As you move your cursor, a tooltip indicates the nearest PolarSnap increment. To restrict point entry to polar distances, both polar tracking and Snap mode (set to PolarSnap) must be on. You can turn off all snapping and tracking temporarily by using an override key.

See also:
■ Override Object Snap Settings on page 230

Quick Reference

DSETTINGS
Sets grid and snap, polar and object snap tracking, object snap modes, and Dynamic Input.

SNAP
Restricts cursor movement to specified intervals.

UNITS
Controls coordinate and angle display formats and precision.

ANGBASE
Sets the base angle to 0 with respect to the current UCS.

ANGDIR
Sets the direction of positive angles.

AUTOSNAP
Controls the display of the AutoSnap marker, tooltip, and magnet.

POLARANG
Sets the polar angle increment.

POLARDIST
Sets the snap increment when the SNAPTYPE is set to 1 (PolarSnap).

POLARMODE
Controls settings for polar and object snap tracking.

SNAPTYPE
Sets the type of snap for the current viewport.
TEMPOVERRIDES

Turns temporary override keys on and off.

TRACKPATH

Controls the display of polar and object snap tracking alignment paths.

Lock an Angle for One Point (Angle)

You can specify an angle override that locks the cursor for the next point entered.

To specify an angle override, enter a left angle bracket (<) followed by an angle whenever a command asks you to specify a point. The Command prompt sequence below shows a 30-degree override entered during a LINE command.

Command: line
Specify first point: Specify a start point for the line
Specify next point or [Undo]: <30
Angle Override: 30
Specify next point or [Undo]: Specify a point

The angle you specify will lock the cursor, overriding Grid Snap, Ortho mode, and PolarSnap. Coordinate entry and object snaps have precedence over an angle override.

Combine or Offset Points and Coordinates

To specify a new point location, you can combine coordinate values from several points or you can specify offsets from existing objects.

Combine Coordinate Values (Coordinate Filters)

You can use coordinate filters to extract one coordinate value at a time from locations on existing objects.

Coordinate filters specify a new coordinate location by using the X value from one location, the Y value of a second location, and, for 3D coordinates, the Z value of a third location. When used with object snaps, coordinate filters extract coordinate values from an existing object.

Coordinate filters are commonly used to locate the center of a rectangle and to locate the projection of a 3D point on the XY plane of the UCS.
To specify a filter at the Command prompt, enter a period and one or more of the letters \( X, Y, \) and \( Z \). The next entry is limited to a specific coordinate value.

**Example: Use of Coordinate Filters in 2D**

In the following illustration, the hole in the holding plate was centered in the rectangle by extracting the \( X,Y \) coordinates from the midpoints of the plate’s horizontal and vertical line segments.

Here is the Command prompt sequence:

Command: `circle`
Specify center point for circle or [3P/2P/Ttr (tangent tangent radius)]: .
\( x \) of: `mid`
of: *Select the horizontal line on the lower edge of the holding plate*
of: (need YZ): `mid`
of: *Select the vertical line on the left side of the holding plate*
of: Diameter/<Radius> *Specify the radius of the hole*

Coordinate filters work only when the program prompts you for a point. If you try to use a coordinate filter at the Command prompt, you see an error message.

**Example: Use of Coordinate Filters in 3D**

This example shows how to use coordinate filters to create a point object at the center (centroid) of a 3D object. Hidden lines have been removed for clarity. The \( X \) value of the new point is extracted from the first location specified, the \( Y \) value from the second location, and the \( Z \) value from the third. The three values are combined to form the coordinate values of the new point.

Command: `point`
Point: .
\( x \) of mid
of select object (1)
(need YZ): .
\( y \)
of mid
of select object (2)
(need Z): `mid`
To use coordinate filters to specify a point in 2D

1. At the prompt for a point, enter a coordinate filter (.x or .y). For example, enter .x to specify the X value first.

2. To extract the first coordinate value, specify a point. For example, if you entered .x in step 1, the X value is extracted from this point.

3. To extract the next coordinate value, specify a different point. The new point location combines the coordinate values extracted from the points you specified in steps 2 and 3.

**NOTE** Instead of specifying a point in steps 2 or 3, you can enter a numeric value.

To use coordinate filters to specify a point in 3D

1. At the prompt for a point, enter a coordinate filter (.x, .y, .z, .xy, .xz, or .yz). For example, enter .x to specify the X value first.

2. To extract the specified coordinate value(s), specify a point. For example, if you entered .x in step 1, the X value is extracted from this point.

3. At the prompt for the remaining coordinates, do one of the following:
   - Extract the remaining coordinate values by specifying a point.
   - Enter another coordinate filter and return to step 2.

   For example, if you entered .x in step 1, specify a second point to extract the Y and Z coordinates simultaneously, or enter .y or .z to specify Y and Z values separately.

   The new point location combines the coordinate values extracted from the points specified in steps 2 and 3.
NOTE Instead of specifying a point in steps 2 or 3, you can enter a numeric value.

Quick Reference

Coordinate Filters (Command Modifier)

Combines X, Y, and Z values from different points to specify a single point.

Track to Points on Objects (Object Snap Tracking)

You can draw objects at specific angles or in specific relationship to other objects along specified directions called alignment paths.

AutoTrack™ helps you draw objects at specific angles or in specific relationships to other objects. When you turn on AutoTrack, temporary alignment paths help you create objects at precise positions and angles. AutoTrack includes two tracking options: polar tracking and object snap tracking.

You can toggle AutoTrack on and off with the Polar and Otrack buttons on the status bar. Use temporary override keys to turn object snap tracking on and off or to turn off all snapping and tracking. See the keyboard illustration in Override Object Snap Settings on page 230.

Object snap tracking works in conjunction with object snaps. You must set an object snap before you can track from an object’s snap point.

Object Snap Tracking

Use object snap tracking to track along alignment paths that are based on object snap points. Acquired points display a small plus sign (+), and you can acquire up to seven tracking points at a time. After you acquire a point, horizontal, vertical, or polar alignment paths relative to the point are displayed as you move the cursor over their drawing paths. For example, you can select a point along a path based on an object endpoint or midpoint or an intersection between objects.

NOTE You can track Perpendicular or Tangent object snap from the last picked point in a command even if the object snap tracking is off.

In the following illustration, the Endpoint object snap is on. You start a line by clicking its start point (1), move the cursor over another line's endpoint (2) to acquire it, and then move the cursor along the horizontal alignment path to locate the endpoint you want for the line you are drawing (3).
Change Object Snap Tracking Settings

By default, object snap tracking is set to orthogonal. Alignment paths are displayed at 0, 90, 180, and 270 degrees from acquired object points. However, you can use polar tracking angles instead. For object snap tracking, object points are automatically acquired.

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.
The direct distance entry method is not available while you are using
the temporary override key for object snap tracking.

Quick Reference

DSETTINGS
Sets grid and snap, polar and object snap tracking, object snap modes, and
Dynamic Input.

OPTIONS
Customizes the program settings.

AUTOSNAP
Controls the display of the AutoSnap marker, tooltip, and magnet.

POLARMODE
Controls settings for polar and object snap tracking.

TRACKPATH
Controls the display of polar and object snap tracking alignment paths.

Track to Offset Point Locations (Tracking)

You can use tracking to specify a point by offsetting vertically and horizontally
from a series of temporary points.

You can use the tracking method whenever you are prompted for a point.
Tracking uses the pointing device to specify a point by offsetting vertically
and horizontally from a series of temporary points. When you start tracking
and specify an initial reference point, the next reference point is constrained
to a path that extends vertically or horizontally from that point. The direction
of the offset is indicated by the rubber-band line. You change the direction
of the offset by moving the cursor through the reference point. You can track
as many points as you need. Typically, you use tracking in combination with
object snaps or direct distance entry.

For example, you can use tracking to find the center point of a rectangle
without using construction lines. Start tracking, and specify the midpoint of
a horizontal line. Drag the cursor vertically and specify the midpoint of a
vertical line (2). Press Enter to accept the point (3) at the center of the rectangle.
Quick Reference

TRACKING (Command Modifier)

Locates a point from a series of temporary points.

Specify Distances

When specifying a point, you can enter distances, offsets, and measured intervals.

Enter Direct Distances

You can specify a point by moving the cursor to indicate a direction and then entering the distance.

To specify a line length quickly, without entering coordinate values, you can specify a point by moving the cursor to indicate a direction and then entering the distance from the first point. You can enter calculated distances and points using the AutoCAD for Mac calculator (CAL).

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 238
- Lock an Angle for One Point (Angle) on page 241
Quick Reference

LINE
   Creates straight line segments.

Direct Distance Entry (Command Modifier)
   Locates the next point at a specified distance in the direction of your cursor.

Offset from Temporary Reference Points
You can establish a temporary reference point as a base point for offsetting subsequent points.

The From command modifier establishes a temporary reference point as a base point for offsetting subsequent points. The From method does not constrain the cursor to orthogonal movement. The From method usually is used in combination with object snaps.

Quick Reference

FROM (Command Modifier)
   Locates a point offset from a reference point within a command.

Specify Intervals on Objects
You can mark off equal distances along objects.

Overview of Specifying Intervals on Objects
Provides a high-level overview of two options for marking off equal distances along objects.

Sometimes you need to create points or insert symbols (blocks) at intervals on an object.

You can
   ■ Specify the length of the segments (MEASURE)
   ■ Specify the number of equal segments (DIVIDE)
You can measure or divide lines, arcs, splines, circles, ellipses, and polylines. With both methods, you can identify the intervals by inserting either a point or a block.

By specifying points, you can use the Node object snap to align other objects at intervals on the measured or divided object. By specifying blocks, you can create precise geometric constructions or insert custom markers. The blocks can rotate at each insertion point.

You cannot insert a block unless it has already been defined within the drawing. Variable attributes within the block are not included when you insert the block references.

The points or blocks you draw using MEASURE or DIVIDE are placed in a selection set. Therefore, if you want to edit them immediately, you can use the Previous option of SELECT.

See also:

- Work with Blocks on page 405

Quick Reference

BLOCK

Creates a block definition from selected objects.

DDPTYPE

Specifies the display style and size of point objects.

DIVIDE

Creates evenly spaced point objects or blocks along the length or perimeter of an object.

MEASURE

Creates point objects or blocks at measured intervals along the length or perimeter of an object.

WBLOCK

Writes objects or a block to a new drawing file.

PDMODE

Controls how point objects are displayed.
PDSIZE

Sets the display size for point objects.

Specify Measured Intervals on Objects

You can mark off equal lengths from one end of a selected object.

You can use MEASURE to mark an object at specified intervals. You can mark the intervals with either points or blocks. The last segment of a measured object may be shorter than the interval you specify.

The starting point for measurements or divisions varies with the object type. For lines or open polylines, the starting point is the endpoint closest to the selection point. For closed polylines, it is the polyline start point. For circles, it is at the angle from the center point that is equivalent to the current snap angle. For example, if the snap angle is 0, the circle starts at the three o’clock position and continues counterclockwise.

If the point marker is displayed as a single dot (the default setting), you may not be able to see the measured intervals. You can change the style of the point markers with the Point Style dialog box (DDPTYPE). The PDMODE system variable also controls the appearance of point markers. For example, you can change the value to make points appear as crosses. PDSIZE controls the size of point objects.

Quick Reference

BLOCK

Creates a block definition from selected objects.

DDPTYPE

Specifies the display style and size of point objects.

MEASURE

Creates point objects or blocks at measured intervals along the length or perimeter of an object.

PDMODE

Controls how point objects are displayed.

PDSIZE

Sets the display size for point objects.
Divide an Object into Equal Segments

You can divide a selected object into a specified number of equal lengths.

You can create points or insert blocks on an object at a specific number of equal intervals. This operation does not actually break an object into individual objects; it only identifies the location of the divisions so that you can use them as geometric reference points.

The starting point for measurements or divisions varies with the object type. For lines or open polylines, the starting point is the endpoint closest to the selection point. For closed polylines, it is the polyline start point. For circles, it is at the angle from the center point that is equivalent to the current snap angle. For example, if the snap angle is 0, the circle starts at the three o'clock position and continues counterclockwise.

If the point marker is displayed as a single dot (the default setting), you may not be able to see the segments. You can change the style of the point markers with the Point Style dialog box (DDPTYPE). The PDMODE system variable also controls the appearance of point markers. For example, you can change the value to make points appear as crosses. PDSIZE controls the size of point objects.

Quick Reference

BLOCK
Creates a block definition from selected objects.

DDPTYPE
Specifies the display style and size of point objects.

DIVIDE
Creates evenly spaced point objects or blocks along the length or perimeter of an object.

WBLOCK
Writes objects or a block to a new drawing file.
PDMODE
Controls how point objects are displayed.

PDSIZE
Sets the display size for point objects.

Extract Geometric Information from Objects
The inquiry and calculation commands can provide information about objects in your drawing and do useful calculations.

Obtain Distances, Angles, and Point Locations
You can obtain information about the relation between two specified points or multiple points; for example, the distance between points or their angle in the XY plane.

To determine the relation between points, you can display the

- Distance between them
- Angle between the points in the XY plane
- Angle of the points from the XY plane
- Delta, or changed, X, Y, and Z distances between them

The ID command lists the X, Y, and Z coordinate values of a specified point.

See also:
- Overview of Coordinate Entry
Quick Reference

DIST
Measures the distance and angle between two points.

ID
Displays the UCS coordinate values of a specified location.

MEASUREGEOM
Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

DISTANCE
Stores the distance computed by the DIST command.

Obtain Area and Mass Properties Information

You can obtain the area, perimeter, and mass properties defined by selected objects or a sequence of points.

You can calculate the area and perimeter of a sequence of points. You can also obtain the area, perimeter, and mass properties of any of several types of objects.

TIP A fast way to calculate an area bounded by several objects in 2D is to use the BOUNDARY command. With BOUNDARY, you can pick a point within the area to create a closed polyline or region. You can then use the Properties Inspector palette or the LIST command to find the area and perimeter of the polyline or region.

Use Commands to Calculate Area

With the MEASUREGEOM and AREA commands, you can specify a series of points or select an object to calculate area. If you need to calculate the combined area of multiple objects, you can keep a running total as you add or subtract one area at a time from the selection set. You cannot use window selection or crossing selection to select objects.

Total area and perimeter are saved in the AREA and PERIMETER system variables.

In addition to area, with the MEASUREGEOM command, you can obtain geometric information from objects such as distance, radius, angle, and volume.
Define an Area

You can measure an arbitrary closed region defined by the points you specify. The points must lie on a plane parallel to the XY plane of the current UCS.

Calculate the Area, Perimeter, or Circumference of an Object

You can calculate the enclosed area and perimeter or circumference of circles, ellipses, polylines, polygons, regions, and AutoCAD 3D solids. The information displayed depends on the type of object selected:

- **Circles.** Area and circumference display.
- **Ellipses, closed polylines, polygons, planar closed spline curves, and regions.** Area and perimeter display. For wide polylines, this area is defined by the center of the width.
- **Open objects such as open spline curves and open polylines.** Area and length display. Area is calculated as though a straight line connects the start point and endpoint.
- **AutoCAD 3D solids.** Total 3D area for the object displays.
Example: How Various Areas Are Calculated

Combined Areas

Calculate Combined Areas
You can calculate the total area of multiple areas by specifying points or by selecting objects. For example, you can measure the total area of selected rooms in a floor plan.

Subtract Areas from Combined Areas
You can subtract more than one area from a combined area as you calculate. For example, if you have calculated the area of a floor plan, you can subtract the area of a room.

Example: Subtraction of Areas from a Calculation
In the following example, the closed polyline represents a metal plate with two large holes. The area of the polyline is first calculated and then the area
of each hole is subtracted. The area and perimeter or circumference of each object displays, with a running total after each step.

The Command prompt sequence is

**Command: area**

Specify first corner point or [Object/Add/Subtract]: a

Specify first corner point or [Object/Subtract]: o

(ADD mode) Select objects: Select the polyline (1)

Area = 0.34, Perimeter = 2.71

Total area = 0.34

(ADD mode) Select objects: Press Enter

Specify first corner point or [Object/Subtract]: s

Specify first corner point or [Object/Add]: o

(SUBTRACT mode) Select objects: Select the lower circle (2)

Area = 0.02, Circumference = 0.46

Total area = 0.32

(SUBTRACT mode) Select objects: Select the upper circle (3)

Area = 0.02, Circumference = 0.46

Total area = 0.30

(SUBTRACT mode) Select circle or polyline: Press Enter

Specify first corner point or [Object/Add]: Press Enter

You can also use REGION to convert the plate and the holes to regions, subtract the holes, and then use the Properties Inspector palette or the LIST command to find the area of the plate.

---

**TIP** Use the CAL command to convert from one system of area units to another.

---

**Calculate Mass Properties**

With the MASSPROP command, you can analyze 3D solids and 2D regions for their mass properties including volume, area, moments of inertia, center of gravity, and so on. In addition, the result of the computations can be saved to a text file.

**See also:**

- [Create and Combine Areas (Regions)](page-289) on page 289
Quick Reference

AREA
Stores the last area computed by the AREA command.

LIST
Displays property data for selected objects.

MASSPROP
Calculates the mass properties of regions or 3D solids.

MEASUREGEOM
Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

PROPERTIES
Controls properties of existing objects.

UNITS
Controls coordinate and angle display formats and precision.

AREA
Stores the last area computed by the AREA command.

PERIMETER
Stores the last perimeter value computed by the AREA or LIST command.

Use a Calculator
You can access a calculator function as you work with the program. You can use the CAL command at the Command prompt transparently while a command is active or not.

Use the Command Prompt Calculator
By entering an expression in the Command prompt calculator, you can quickly solve a mathematical problem or locate points in your drawing.
The CAL command runs the 3D calculator utility to evaluate vector expressions (combining points, vectors, and numbers) and real and integer expressions. The calculator performs standard mathematical functions. It also contains a set of specialized functions for calculations involving points, vectors, and AutoCAD for Mac 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.

<table>
<thead>
<tr>
<th>Operator</th>
<th>Operation</th>
</tr>
</thead>
<tbody>
<tr>
<td>( )</td>
<td>Groups expressions</td>
</tr>
<tr>
<td>^</td>
<td>Indicates numeric exponent</td>
</tr>
<tr>
<td>*, /</td>
<td>Multiplies and divides numbers</td>
</tr>
<tr>
<td>+, -</td>
<td>Adds and subtracts numbers</td>
</tr>
</tbody>
</table>

### Calculating Points
You can use CAL whenever you need to calculate a point or a number within a command.

For example, you enter $(\text{mid}+\text{cen})/2$ to specify a point halfway between the midpoint of a line and the center of a circle.
The following example uses CAL as a construction tool. It locates a center point for a new circle, and then calculates one fifth of the radius of an existing circle.

Here is the command prompt sequence:

Command: circle
Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: 'cal
>> Expression: \((\text{mid}+\text{cen})/2\)
>> Select entity for MID snap: Select the notch line (1)
>> Select entity for CEN snap: Select the large circle (2)
Diameter/<Radius>: 'cal
>> Expression: \(1/5*\text{rad}\)
>> Select circle, arc or polyline segment for RAD function: Select the large circle (3)

To start the Command prompt calculator
Do one of the following:

■ At the Command prompt, enter CAL. Then, enter a CAL expression.
■ At a prompt for a command in progress, enter ‘CAL to start the CAL command transparently. Then, enter a CAL expression to calculate a value for that prompt.

Quick Reference

CAL
Evaluates mathematical and geometric expressions.
Draw Geometric Objects

You can create a range of objects, from simple lines and circles to spline curves, and ellipses. In general, you draw objects by specifying points with the pointing device or by entering coordinate values at the Command prompt.

Draw Linear Objects

A line, the most basic object, can be one segment or a series of connected segments.

Draw Lines

You can close a sequence of line segments so that the first and last segments are joined.

You can assign properties to lines including color, linetype, and lineweight. For more information about properties, see Work with Object Properties on page 161.

You specify the locations that define the endpoints of each line with precision. You can

- Enter the coordinate values for an endpoint, using either absolute or relative coordinates
- Specify an object snap relative to an existing object. For example, you can specify the center of a circle as one endpoint of the line
- Turn grid snap on and snap to a location

There are other methods for creating precise lines. A highly efficient technique is to offset a line from an existing line, and then trim or extend it to the desired length.
Use polyline objects instead of line objects if you want the segments to be connected as a single object.

See also:
- Enter Coordinates to Specify Points
- Use Object Snaps on page 226
- Adjust Grid and Grid Snap on page 232
- Draw Polylines on page 262
- Offset an Object on page 333
- Break and Join Objects on page 351
- Infer Geometric Constraints on page 384

Quick Reference

Commands

LINE
- Creates straight line segments.

RAY
- Creates a line that starts at a point and continues to infinity.

XLINE
- Creates a line of infinite length.

Draw Polylines

A polyline is a connected sequence of segments created as a single object. You can create straight line segments, arc segments, or a combination of the two.
Polylines are ideal for applications including the following:

- Contour lines for topographic, isobaric, and other scientific applications
- Wiring diagrams and printed circuit board layouts
- Process and piping diagrams
- Extrusion profiles and extrusion paths for 3D solid modeling

Polylines can be created with several commands including PLINE, RECTANG, POLYGON, DONUT, BOUNDARY, and REV CLOUD. All of these commands result in a LW POLYLINE (lightweight polyline) object type.

With the 3DPOLY command, you can create non-planar polylines that result in a POLYLINE object type. Fewer options are available with 3D polylines.

After you create a polyline, you can edit it using grips or PEDIT. You can use EXPLODE to convert polylines to individual line and arc segments.

**NOTE** You can convert a spline-fit polyline created with PEDIT into a true spline object with SPLINE.

**Create Wide Polylines**

You can draw polylines of various widths by using the Width and Halfwidth options. You can set the width of individual segments and make them taper gradually from one width to another. These options become available after you specify a starting point for the polyline.

The Width and Halfwidth options set the width of the next polyline segments you draw. Widths greater than zero produce wide lines, which are filled if Fill mode is on and outlined if Fill mode is off.

Intersections of adjacent wide segments are usually beveled. However, nontangent arc segments, acute angles, or segments that use a dash-dot linetype are not beveled.
Create Polylines from the Boundaries of Objects

You can create a polyline from the boundaries of objects that form a closed area with BOUNDARY. A polyline created using this method is a separate object, distinct from the objects used to create it.

To expedite the boundary selection process in large or complex drawings, you can specify a group of boundary candidates, called a boundary set. You create this set by selecting the objects you want to use define the boundary.

See also:
- Draw Rectangles and Polygons on page 265
- Modify Complex Objects on page 353
- Break and Join Objects on page 351
- Control Lineweights on page 186
- Infer Geometric Constraints on page 384

Quick Reference

Commands

3DPOLY
- Creates a 3D polyline.

BOUNDARY
- Creates a region or a polyline from an enclosed area.

EXPLODE
- Breaks a compound object into its component objects.

FILL
- Controls the filling of objects such as hatches, 2D solids, and wide polylines.
PEDIT
Edits polylines and 3D polygon meshes.

PLINE
Creates a 2D polyline.

POLYGON
Creates an equilateral closed polyline.

RECTANG
Creates a rectangular polyline.

System Variables

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

HPBOUND
Controls the object type created by HATCH and BOUNDARY.

PLINECONVERTMODE
Specifies the fit method used in converting splines to polylines.

PLINEGEN
Sets how linetype patterns generate around the vertices of a 2D polyline.

PLINETYPE
Specifies whether optimized 2D polylines are used.

PLINEWID
Stores the default polyline width.

Draw Rectangles and Polygons

You can create rectangles and regular polygons quickly. Creating polygons is a simple way to draw equilateral triangles, squares, pentagons, hexagons, and so on.

If necessary, you can use EXPLODE to convert the resulting polyline object into lines.
**Draw Rectangles**

Use RECTANG to create closed polylines in a rectangular shape.

**Draw Regular Polygons**

Use POLYGON to create closed polylines with between 3 and 1,024 equal-length sides. The following illustrations show polygons created using three methods. In each case, two points are specified.

See also:
- Draw Polylines on page 262
- Infer Geometric Constraints on page 384

**Quick Reference**

**Commands**

**BOUNDARY**

Creates a region or a polyline from an enclosed area.

**EXPLODE**

Breaks a compound object into its component objects.

**POLYGON**

Creates an equilateral closed polyline.

**RECTANG**

Creates a rectangular polyline.
System Variables

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

HPBOUND
Controls the object type created by HATCH and BOUNDARY.

PLINEWID
Stores the default polyline width.

POLYSIDES
Sets the default number of sides for the POLYGON command.

SNAPANG
Sets the snap and grid rotation angle for the current viewport relative to the current UCS.

Draw Multiline Objects

Multilines are composed of parallel lines, called elements.

When you draw a multiline, you can use the STANDARD style, which has two elements, or specify a style that you created previously. You can also change the justification and scale of the multiline before you draw it.

Multiline justification determines which side of the cursor that the multiline is drawn, or whether it is centered on the cursor.

Multiline scale controls the overall width of the multiline using the current units. Multiline scale does not affect linetype scale. If you change the multiline scale, you might need to make equivalent changes to the linetype scale to prevent dots or dashes from being disproportionately sized.

Create Multiline Styles

You can create named styles for multilines to control the number of elements and the properties of each element. The properties of multilines include

- The total number of elements and position of each element
- The offset distance for each element from the middle of the multiline
- The color and linetype of each element
The visibility of the lines, called *joints*, that appear at each vertex

- The type of end caps that are used
- The background fill color of the multiline

Elements with a positive offset appear on one side of the middle of the multiline; elements with a negative offset appear on the other side of the middle of the multiline.

**See also:**
- *Modify Multilines* on page 364

**Quick Reference**

**Commands**

**OFFSET**

- Creates concentric circles, parallel lines, and parallel curves.

**MLINE**

- Creates multiple parallel lines.

**MLSTYLE**

- Creates, modifies, and manages multiline styles.

**System Variables**

**CMLSTYLE**

- Sets the multiline style that governs the appearance of the multiline.
Draw Freehand Sketches

Sketching is useful for creating irregular boundaries or for tracing with a digitizer.

Draw freehand sketches with the SKETCH command. Freehand sketches comprise many line segments that are converted into a line, polyline, or spline.

For Splines, you can determine how closely the spline's curve fits to the freehand sketch.

For any sketch type, set the minimum length (increment) of the line segments. Small line segments allow greater accuracy, but they can greatly increase the drawing file size.

Before sketching, check the CELTYPE system variable to make sure the current linetype is BYLAYER. When you sketch with dot or dash linetypes, smaller line segments can become invisible.

Quick Reference

Commands
SKETCH
Creates a series of freehand line segments.

System Variables
SKETCHINC
Sets the record increment for the SKETCH command.
SKPOLY
Determines whether the SKETCH command generates lines, polylines, or splines.
Draw Curved Objects

Curved objects are arcs, circles, polyline arcs, donuts, ellipses, and splines.

Draw Arcs

To create an arc, you can specify various combinations of center, endpoint, start point, radius, angle, chord length, and direction values.

You can create arcs in several ways. With the exception of the first method, arcs are drawn counterclockwise from the start point to the endpoint.

Draw Arcs by Specifying Three Points

You can create an arc by specifying three points. In the following example, the start point of the arc snaps to the endpoint of a line. The second point of the arc snaps to the middle circle in the illustration.

Draw Arcs by Specifying Start, Center, End

You can create an arc using a start point, center, and a third point that determines the endpoint.

The distance between the start point and the center determines the radius. The endpoint is determined by a line from the center that passes through the third point. The resulting arc is always created counterclockwise from the start point.

Using different options, you can specify either the start point first or the center point first.
Draw Arcs by Specifying Start, Center, Angle

You can create an arc using a start point, center, and an included angle. The distance between the start point and the center determines the radius. The other end of the arc is determined by specifying an included angle that uses the center of the arc as the vertex. The resulting arc is always created counterclockwise from the start point.

Using different options, you can specify either the start point first or the center point first.

The included angle determines the endpoint of the arc. Use the Start, End, Angle method when you know both endpoints but cannot snap to a center point.

Draw Arcs by Specifying Start, Center, Length

You can create an arc using a start point, center, and the length of a chord. The distance between the start point and the center determines the radius. The other end of the arc is determined by specifying the length of a chord.
between the start point and the endpoint of the arc. The resulting arc is always created counterclockwise from the start point.

Using different options, you can specify either the start point first or the center point first.

The length of the chord of the arc determines the included angle.

**Draw Arcs by Specifying Start, End, Angle**

You can create an arc using a start point, endpoint, and an included angle. The included angle between the endpoints of the arc determines the center and the radius of the arc.

**Draw Arcs by Specifying Start, End, Direction**

You can create an arc using a start point, endpoint, and a tangent direction at the start point.

The tangent direction can be specified either by locating a point on the desired tangent line, or by entering an angle. You can determine which endpoint controls the tangent by changing the order in which you specify the two endpoints.

**Draw Arcs by Specifying Start, End, Radius**

You can create an arc using a start point, endpoint, and a radius. The direction of the bulge of the arc is determined by the order in which you specify its endpoints. You can specify the radius either by entering it or by specifying a point at the desired radius distance.
Draw Contiguous Tangent Arcs and Lines

Immediately after you create an arc, you can start a line that is tangent to the arc at an endpoint by starting the LINE command and pressing Enter at the Specify First Point prompt. You need to specify only the line length.

Immediately after you create a line or an arc, you can start an arc that is tangent at an endpoint by starting the ARC command and pressing Enter at the Specify Start Point prompt. You need to specify only the endpoint of the new arc.

See also:
- Draw Polylines on page 262
- Break and Join Objects on page 351

Quick Reference

Commands

ARC
Creates an arc.

LINE
Creates straight line segments.

OFFSET
Creates concentric circles, parallel lines, and parallel curves.
VIEWRES
Sets the resolution for objects in the current viewport.

System Variables
ANGDIR
Sets the direction of positive angles.
LASTANGLE
Stores the end angle of the last arc entered relative to the XY plane of the current UCS for the current space.
WHIPARC
Controls whether the display of circles and arcs is smooth.

Draw Circles
To create circles, you can specify various combinations of center, radius, diameter, points on the circumference, and points on other objects.

You can create circles in several ways. The default method is to specify the center and the radius. Three other ways to draw a circle are shown in the illustration.

Draw a Circle Tangent to Other Objects
The tangent point is a point where an object touches another object without intersecting it. To create a circle that is tangent to other objects, select the objects and then specify the radius of the circle. In the illustrations below, the bold circle is the one being drawn, and points 1 and 2 select the objects to which it is tangent.
To create a circle tangent at three points, set running object snaps (OSNAP) to Tangent and use the three-point method to create the circle.

See also:
- Use Object Snaps on page 226
- Draw Isometric Circles on page 953

Quick Reference

Commands
CIRCLE
  Creates a circle.
OFFSET
  Creates concentric circles, parallel lines, and parallel curves.

System Variables
CIRCLERAD
  Sets the default circle radius.
WHIPARC
  Controls whether the display of circles and arcs is smooth.

Draw Polyline Arcs
A polyline is a connected sequence of line segments created as a single object. You can create straight line segments, arc segments, or a combination of the two.
Multisegmented lines provide editing capabilities unavailable for single lines. For example, you can adjust their width and curvature. After you've created a polyline, you can edit it with PEDIT or use EXPLODE to convert it to individual line and arc segments. You can

- Convert a spline-fit polyline into a true spline with SPLINE
- Use closed polylines to create a polygon
- Create a polyline from the boundaries of overlapping objects

**Create Arc Polylines**

When you draw arc segments in a polyline, the first point of the arc is the endpoint of the previous segment. You can specify the angle, center point, direction, or radius of the arc. You can also complete the arc by specifying a second point and an endpoint.

**Create Closed Polylines**

You can draw a closed polyline to create a polygon. To close a polyline, specify the starting point of the last side of the object, enter c (Close), and press Enter.

**Create Wide Polylines**

You can draw polylines of various widths by using the Width and Halfwidth options. You can set the width of individual segments and make them taper gradually from one width to another. These options become available after you specify a starting point for the polyline.

The Width and Halfwidth options set the width of the next polyline segments you draw. Zero (0) width produces a thin line. Widths greater than zero produce wide lines, which are filled if Fill mode is on and outlined if Fill mode.
is off. The Halfwidth option sets width by specifying the distance from the center of the wide polyline to an outside edge.

**Taper**

When you use the Width option, you are prompted for both a starting and an ending width. By entering different values, you can taper the polyline. The starting and ending points of wide polyline segments are in the center of the line. Intersections of adjacent wide segments are usually beveled. However, nontangent arc segments, acute angles, or segments that use a dash-dot linetype are not beveled.

**Create Polylines from the Boundaries of Objects**

You can create a polyline from the boundaries of overlapping objects that form a closed area. A polyline created using the boundary method is a separate object, distinct from the objects used to create it. You can edit it using the same methods used to edit other polylines.

To expedite the boundary selection process in large or complex drawings, you can specify a group of boundary candidates, called a boundary set. You create this set by selecting the objects you want to use to define the boundary.

---

See also:

- Modify Splines on page 359
- Modify Polylines on page 355
- Break and Join Objects on page 351
- Control Lineweights on page 186
- Infer Geometric Constraints on page 384
Quick Reference

Commands

3DPOLY
Creates a 3D polyline.

BOUNDARY
Creates a region or a polyline from an enclosed area.

OFFSET
Creates concentric circles, parallel lines, and parallel curves.

PEDIT
Edits polylines and 3D polygon meshes.

PLINE
Creates a 2D polyline.

VIEWRES
Sets the resolution for objects in the current viewport.

System Variables

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

HPBOUND
Controls the object type created by HATCH and BOUNDARY.

PLINECONVERTMODE
Specifies the fit method used in converting splines to polylines.

PLINEGEN
Sets how linetype patterns generate around the vertices of a 2D polyline.

PLINETYPE
Specifies whether optimized 2D polylines are used.

PLINEWID
Stores the default polyline width.
**Draw Donuts**

Donuts are filled rings or solid-filled circles that actually are closed polylines with width.

To create a donut, you specify its inside and outside diameters and its center. You can continue creating multiple copies with the same diameter by specifying different center points. To create solid-filled circles, specify an inside diameter of 0.

---

**Quick Reference**

**Commands**

DONUT

Creates a filled circle or a wide ring.

FILL

Controls the filling of objects such as hatches, 2D solids, and wide polylines.

**System Variables**

DONUTID

Sets the default for the inside diameter of a donut.

DONUTOD

Sets the default for the outside diameter of a donut.

FILLMODE

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

The shape of an ellipse is determined by two axes that define its length and width. The longer axis is called the major axis, and the shorter one is the minor axis.

The illustrations below show two different ellipses created by specifying axis and distance. The third point specifies only a distance and does not necessarily designate the axis endpoint.

If you are drawing on isometric planes to simulate 3D, you can use ellipses to represent isometric circles viewed from an oblique angle. First you need to turn on Isometric Snap in the Drafting Settings dialog box (DSETTINGS command).

**See also:**
- [Draw Isometric Circles](#) on page 953
- [Break and Join Objects](#) on page 351
Quick Reference

Commands
ELLIPSE
   Creates an ellipse or an elliptical arc.

System Variables
ANGDIR
   Sets the direction of positive angles.
PELLIPSE
   Controls the ellipse type created with ELLIPSE.

Draw Splines
A spline is a smooth curve that passes through or near a set of points that influence the shape of the curve.

The SPLINE command creates curves called Non-Uniform Rational B-Splines (NURBS), which we will refer to as splines for simplicity.

By default, a spline is a blend of curve segments of degree 3 (cubic) polynomials. Cubic splines are the most commonly used ones, and they correspond with splines created manually using thin wooden strips shaped by weights at data points.

For example, the SPLINE command was used to create one boundary of the concrete walk that extends between the driveway and the house as shown.
Splines are also used for creating solids and surfaces for 3D modeling. For more information, see Create Solids and Surfaces from Lines and Curves on page 446.

Understand Control Vertices and Fit Points

You can create or edit splines using either control vertices (CVs), or fit points. The spline on the left displays control vertices along a control polygon, and the spline on the right displays fit points.

The options available in the SPLINE command depend on which method is used to create the spline.

The CVSHOW and CVHIDE commands determine whether the control vertices are displayed on a spline even when the spline is not selected.

Use the triangular grip on a selected spline to switch between displaying control vertices and displaying fit points. You can use the round and square grips to modify a selected spline. For more information, see Modify Splines on page 359.

IMPORTANT Switching from displaying control vertices to fit points automatically changes the selected spline to degree 3. Splines originally created using higher-degree equations will likely change shape as a result.

Create Splines Using Control Vertices

When you create splines using control vertices, the points you specify display temporary lines between them, forming a control polygon that determines the shape of the spline.
Once the spline is created, many people prefer changing the shape of the spline using control vertices because of the fine control this method provides. With this method, you can also specify lower or higher degree polynomials, including degree 1 (linear), degree 2 (quadratic), degree 3 (cubic), and so on up to degree 10.

**Create Splines Using Fit Points**

When you create splines using fit points, the resulting curve passes through the specified points, and is influenced by the spacing of mathematical *knots* in the curve.

You can choose the spacing of these knots with the *knot parameterization* option, which will result in different curves as shown in the example.

![Example of fit points and knots](image)

**NOTE** There is no best choice for knot parameterization for all cases. The chord length parameterization is commonly used, and the square root (centripetal) parameterization often produces better curves depending on the data set.

When the Tolerance value is set to 0, the spline passes directly through the fit points. With larger tolerance values, the spline passes near the fit points. Optionally, you can specify the tangent direction for the spline at each end.

**NOTE** The fit point method always results in a degree 3 spline.

**Special Cases**

You can create a spline with a parabolic shape by specifying a degree 2 spline created with exactly 3 control vertices as shown on the left. Degree 3 splines created with 4 control vertices have the same shape as Bezier curves of degree 3 as shown on the right.
You can close a spline so that the start point and end point are coincident and tangent (C1 continuity).

**NOTE** Because periodic curves are not currently supported, closed splines and 3D surfaces created from closed splines may kink when reshaped.

A legacy method for creating approximations of B-splines involves creating a polyline, and then using the Spline option of the PEDIT command to generate a spline-fit polyline.

**See also:**
- **Modify Splines** on page 359
- **Break and Join Objects** on page 351
- **Create Solids and Surfaces from Lines and Curves** on page 446

**Quick Reference**

**Commands**

**CVHIDE**

Turns off the display of control vertices for all NURBS surfaces and curves.

**CVSHOW**

Displays the control vertices for specified NURBS surfaces or curves.
PEDIT
Edits polylines and 3D polygon meshes.

PLINE
Creates a 2D polyline.

SPLINE
Creates a smooth B-spline curve that passes through or near a set of points that controls the shape of the curve.

SPLINEDIT
Modifies the parameters of a spline or converts a spline-fit polyline to a spline.

System Variables
PLINECONVERTMODE
Specifies the fit method used in converting splines to polylines.

Draw Helixes
A helix is an open 2D or 3D spiral.

You can use a helix as a path with the SWEEP command. For example, you might sweep a circle along a helix path to create a solid model of a spring.

When you create a helix, you can specify the following:

■ Base radius
■ Top radius
■ Height
■ Number of turns
■ Turn height
■ Twist direction

If you specify the same value for both the base radius and the top radius, then a cylindrical helix is created. By default, the top radius is set to the same value as the base radius. You cannot specify 0 for both the base radius and top radius.
If you specify different values for the top radius and the base radius, then a conical helix is created. If you specify a height value of 0, then a flat, 2D spiral is created.

**NOTE** A helix is a spline approximation of a real helix. Length values may not be completely accurate. However, when you use a helix as a sweep path, the resulting values will be accurate regardless of the approximation.

See also:
- Modify Helixes on page 363
- Create a Solid or Surface by Sweeping on page 453

**Quick Reference**

**Commands**
HELIX

Creates a 2D spiral or 3D spring.

**Draw Construction and Reference Geometry**

Construction lines and reference points are temporary objects you create to help you draw accurately.

**Draw Reference Points**

Point objects are useful as nodes or reference geometry for object snaps and relative offsets.

You can set the style of the points and their size relative to the screen or in absolute units. Changing the style of points

- Makes them more visible and easier to differentiate from grid dots
- Affects the display of all point objects in the drawing
- Requires using REGEN to make the change visible
Quick Reference

Commands

DDPTYPE
Specifies the display style and size of point objects.

POINT
Creates a point object.

System Variables

PDMODE
Controls how point objects are displayed.

PDSIZE
Sets the display size for point objects.

Draw Construction Lines (and Rays)

Lines that extend to infinity in one or both directions, known as rays and construction lines, respectively, can be used as references for creating other objects.

For example, you can use construction lines to find the center of a triangle, prepare multiple views of the same item, or create temporary intersections to use for object snaps.

Infinite lines do not change the total area of the drawing. Therefore, their infinite dimensions have no effect on zooming or viewpoints, and they are ignored by commands that display the drawing extents. You can move, rotate, and copy infinite lines just as you can move, rotate, and copy other objects. You may want to create infinite lines on a construction line layer that can be frozen or turned off before plotting.

Construction Lines

A construction line (also known as xlines) can be placed anywhere in three-dimensional space. You can specify its orientation in several ways. The default method for creating the line is the two-point method: you specify two points to define the orientation. The first point, the root, is the conceptual midpoint of the construction line, that is, the point snapped to by the Midpoint object snap.
You can also create construction lines in several other ways.

- **Horizontal and Vertical.** Create construction lines that pass through a point you specify and are parallel to the X or Y axis of the current UCS.

- **Angle.** Creates a construction line in one of two ways. Either you select a reference line and then specify the angle of the construction line from that line, or you create a construction line at a specific angle to the horizontal axis by specifying an angle and then a point through which the construction line should pass.

- **Bisector.** Creates a construction line that bisects an angle you specify. You specify the vertex and the lines that create the angle.

- **Offset.** Creates a construction line parallel to a baseline you specify. You specify the offset distance, select the baseline, and then indicate on which side of the baseline to locate the construction line.

**Rays**

A ray is a line in three-dimensional space that starts at a point you specify and extends to infinity. Unlike construction lines, which extend in two directions, rays extend in only one direction. Using rays instead of construction lines can help reduce visual clutter. Like construction lines, rays are ignored by commands that display the drawing extents.

**Quick Reference**

**Commands**

RAY

Creates a line that starts at a point and continues to infinity.
XLINE

Creates a line of infinite length.

Create and Combine Areas (Regions)

Regions are 2D enclosed areas that have physical properties such as centroids or centers of mass. You can combine existing regions into a single, complex region.

Regions can be used for

- Extracting design information, such as areas and centroids, using MASSPROP
- Applying hatching and shading
- Combining simple objects into more complex ones with Boolean operations.

You can create regions from objects that form closed loops. Loops can be combinations of lines, polylines, circles, arcs, ellipses, elliptical arcs, and splines that enclose an area.

You create regions using the REGION command to convert a closed object into a region, and the BOUNDARY command to create a region from an area enclosed by objects. You can combine regions by unifying, subtracting, or intersecting them.

Objects combined using UNION:
Objects combined using SUBTRACT:

Objects combined using INTERSECT:

Invalid Boundaries

When a boundary cannot be determined, it might be because the specified internal point is not within a fully enclosed area. With the BOUNDARY command, red circles are displayed around unconnected endpoints of the boundary to identify gaps in the boundary.

The red circles remain displayed even after you exit the command. They are removed when you specify a closed boundary, or by using REDRAW, REGEN, or REGENALL.
Quick Reference

Commands
BOUNDARY
Creates a region or a polyline from an enclosed area.
INTERSECT
Creates a 3D solid, surface, or 2D region from overlapping solids, surfaces, or regions.
MASSPROP
Calculates the mass properties of regions or 3D solids.
REGION
Converts an object that encloses an area into a region object.
SUBTRACT
Combines selected 3D solids or 2D regions by subtraction.
UNION
Combines selected 3D solids, surfaces, or 2D regions by addition.

System Variables
DELOBJ
Controls whether geometry used to create 3D objects is retained or deleted.

Create Revision Clouds
Revision clouds are polylines that consist of sequential arcs. They are used to call attention to parts of a drawing during the review stage.
If you review or redline drawings, you can increase your productivity by using the Revision Cloud feature to highlight your markups. REVCL OUD creates a polyline of sequential arcs to form a cloud-shaped object. You can select a style for a revision cloud: Normal or Calligraphy. If you select Calligraphy, the revision cloud looks as if it was drawn with a calligraphy pen.
You can create a revision cloud from scratch, or you can convert objects, such as a circle, ellipse, polyline, or spline, to a revision cloud. When you convert an object to a revision cloud, the original object is deleted if DELOBJ is set to 1 (the default).
You can set the minimum and maximum default values for the arc lengths of a revision cloud. When you draw a revision cloud, you can vary the size of the arcs by using pick points for the smaller arc segments. You can also edit the individual arc lengths and chord lengths of a revision cloud by adjusting the pick points.

REVCLOUD stores the last used arc length as a multiple of the DIMSCALE system variable to provide consistency among drawings with different scale factors.

Make sure that you can see the entire area to be outlined with REVCLOUD before you begin the command. REVCLOUD is not designed to support transparent and real-time panning and zooming.

**Quick Reference**

**Commands**

REVCLOUD

Creates a revision cloud using a polyline.

**System Variables**

DIMSCALE

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

DELOBJ

Controls whether geometry used to create 3D objects is retained or deleted.
Change Existing Objects

You can select objects, view and edit object properties, and perform general and object-specific editing operations.

Select Objects

You have a wide range of options when you need to select objects for editing operations.

Select Objects Individually

At the Select Objects prompt, you can select one or more objects individually.

Use the Pickbox Cursor

When the square pickbox cursor is in position to select an object, the object is highlighted. Click to select the object.

You can control the size of the pickbox in the Application Preferences dialog box, Cursor & Selection tab.

Select Objects Close Together

It is difficult to select objects that are close together or lie directly on top of one another. The example shows two lines and a circle that lie within the pickbox.
If selection preview is turned on, you can cycle through the objects by rolling over the object on top to highlight it, and pressing and holding Shift and then pressing Spacebar continuously. When the required object is highlighted, left-click to select it.

If selection preview is turned off, hold down Shift+Spacebar and click to cycle through these objects, one after the other, until the one you want is selected. Press Esc to turn off cycling.

**Remove Selection from Objects**
Remove objects from the current selection set by holding down Shift and selecting them again.

**See also:**
- Modify 3D Subobjects on page 570
- Modify Composite Solids and Surfaces on page 583

**Quick Reference**

**PROPERTIES**
- Controls properties of existing objects.

**SELECT**
- Places selected objects in the Previous selection set.

**3DSELECTIONMODE**
- Controls the selection precedence of both visually and physically overlapping objects when using 3D visual styles.

**HIGHLIGHT**
- Controls object highlighting; does not affect objects selected with grips.

**LEGACYCTRLPICK**
- Specifies the keys for selection cycling and the behavior for Ctrl-click.

**PICKADD**
- Controls whether subsequent selections replace the current selection set or add to it.
PICKAUTO
Controls automatic windowing at the Select Objects prompt.

PICKBOX
Sets the object selection target height, in pixels.

PICKDRAG
Controls the method of drawing a selection window.

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

Select Multiple Objects
At the Select Objects prompt, you can select many objects at the same time.

Specify a Rectangular Selection Area
Specify opposite corners to define a rectangular area. The background inside the area changes color and becomes transparent. The direction that you drag your cursor from the first point to the opposite corner determines which objects are selected.

- **Window selection.** Drag your cursor from left to right to select only objects that are entirely enclosed by the rectangular area.

- **Crossing selection.** Drag your cursor from right to left to select objects that the rectangular window encloses or crosses.
With a window selection, usually the entire object must be contained in the rectangular selection area. However, if an object with a noncontinuous (dashed) linetype is only partially visible in the viewport and all the visible vectors of the linetype can be enclosed within the selection window, the entire object is selected.

**Specify an Irregularly Shaped Selection Area**

Specify points to define an irregularly shaped area. Use window polygon selection to select objects entirely enclosed by the selection area. Use crossing polygon selection to select objects enclosed or crossed by the selection area.

**Specify a Selection Fence**

In a complex drawing, use a selection fence. A selection fence looks like a polyline and selects only the objects it passes through. The circuit board illustration shows a fence selecting several components.
Use Other Selection Options

You can see all selection options by entering ? at the Select Objects prompt. For a description of each of the selection options, see SELECT.

Remove Selection from Multiple Objects

You can enter r (Remove) at the Select Objects prompt and use any selection option to remove objects from the selection set. If you are using the Remove option and want to return to adding objects to the selection set, enter a (Add).

You can also remove objects from the current selection set by holding down Shift and selecting them again, or by holding down Shift and then clicking and dragging window or crossing selections. You can add and remove objects repeatedly from the selection set.

Quick Reference

PROPERTIES
Controls properties of existing objects.

QSELECT
Creates a selection set based on filtering criteria.

SELECT
Places selected objects in the Previous selection set.

HIGHLIGHT
Controls object highlighting; does not affect objects selected with grips.

PICKADD
Controls whether subsequent selections replace the current selection set or add to it.
PICKAUTO
Controls automatic windowing at the Select Objects prompt.

PICKBOX
Sets the object selection target height, in pixels.

PICKDRAG
Controls the method of drawing a selection window.

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

PREVIEWEFFECT
Specifies the visual effect used for previewing selection of objects.

Prevent Objects from Being Selected
You can prevent objects on specified layers from being selected and modified by locking those layers.

Typically, you lock layers to prevent accidental editing of particular objects. Other operations are still possible when a layer is locked. For example, you can make a locked layer current, and you can add objects to it. You can also use inquiry commands (such as LIST), use object snaps to specify points on objects on locked layers, and change the draw order of objects on locked layers.

To help you differentiate between locked and unlocked layers, you can do the following:

■ Hover over an object to see whether a lock icon is displayed
■ Dim the objects on locked layers

**NOTE** Grips are not displayed on objects that are on locked layers.

Quick Reference

LAYER
Manages layers and layer properties.
LAYISO

Hides or locks all layers except those of the selected objects.

LAYLCK

Locks the layer of a selected object.

LAYULK

Unlocks the layer of a selected object.

LAYLOCKFADECTL

Controls the amount of fading for objects on locked layers.

Select Objects by Properties

Use object properties or object types to include objects in a selection set, or to exclude them.

To quickly define a selection set based on specified filtering criteria, use

- Select Similar (SELECTSIMILAR) to select similar objects of the same type based on specified matching properties
- AutoLISP function SSGET to create a custom selection filter

If you added a feature classification to an object using Autodesk or a third-party application, you can select objects by classification property. With object selection filters, you can name and save filters for future use.

With object selection filters, to filter your selection set based on color, linetype, or lineweight, consider whether these properties are set to BYLAYER for any objects in your drawing. For example, an object may appear red because its color is set to BYLAYER and the layer color is red.

By default, objects of the same type are considered similar if they are on the same layer, and, for blocks and other referenced objects, have the same name. Subobjects are only considered at the object level. For example, when a mesh vertex is selected, SELECTSIMILAR selects other mesh objects, not just the mesh vertices.

See also:

- Customize Object Selection on page 300
- Work with Layers on page 165
Quick Reference

PROPERTIES
- Controls properties of existing objects.

SELECT
- Places selected objects in the Previous selection set.

SELECTSIMILAR
- Adds similar objects to the selection set based on selected objects.

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

PICKAUTO
- Controls automatic windowing at the Select Objects prompt.

PICKBOX
- Sets the object selection target height, in pixels.

PICKDRAG
- Controls the method of drawing a selection window.

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

SELECTSIMILARMODE
- Controls which properties must match for an object of the same type to be selected with SELECTSIMILAR.

Customize Object Selection

You can control several aspects of selecting objects, such as whether you enter a command first or select objects first, the size of the pickbox cursor, and how selected objects are displayed.

For commands that use the Select Objects prompt, you can

- Enter a command first, and then select objects
- Select the objects first, and then enter a command
You can also choose

- Whether objects to be selected are previewed during selection
- Whether selected objects are highlighted
- How you define selection areas and how you create selection sets

**Select the Command First**

When you use an editing command, a Select Objects prompt is displayed and the crosshairs is replaced with a pickbox. You can respond to the Select Objects prompt in various ways:

- Select objects one at a time.
- Click an empty area. Drag the cursor to define a rectangular selection area.
- Enter a selection option. Enter `?` to display all selection options.
- Combine selection methods. For example, to select most of the objects in the drawing area, select all objects and then remove the objects that you do not want selected.

**Select Objects First**

You can use one of two methods to select objects before starting a command:

- Use the `SELECT` command, and enter `?` to display all selection options. All objects selected are put into the Previous selection set. To use the Previous selection set, enter `p` at the Select Objects prompt of any subsequent command.

- When noun/verb selection is turned on, select objects at the Command prompt before entering a command such as MOVE, COPY, or ERASE. With this method, you can only select objects by clicking them individually or by using automatic selection.

**Highlight Objects to Be Selected**

Objects are highlighted when the pickbox cursor rolls over them, providing a preview of which object will be selected when you click. When you specify an area to select multiple objects, the background of the area becomes transparent.
These selection previewing effects are turned on by default. You can turn them off with the SELECTIONPREVIEW system variable. When the PICKBOX system variable is set to 0, selection previewing of objects is not available.

**Control the Appearance of Selected Objects**

By default, selected objects are displayed with dashed lines. You can increase program performance by setting the HIGHLIGHT system variable to 0. Turning off selection highlighting does not affect grips on selected objects.

**Set Up Default Selection Methods**

The default selection methods are:

- Use selection previewing and selection area effects to preview selection.
- Select objects before entering a command (noun-verb selection) or after entering a command. (PICKFIRST)
- Press Shift to append objects to the selection set. (PICKADD)
- Click and drag to create a selection window. Otherwise you must click twice to define the corners of a selection window. (PICKDRAG)
- Start Window or Crossing selection automatically when you click an empty space. Otherwise, you must enter c or w to specify window crossing selection. (PICKAUTO)
- Change the size of the pickbox. (PICKBOX)
- Select all objects in a group when you select one object in that group.
- Include the boundary in the selection set when you select a hatch.

**Quick Reference**

**PROPERTIES**

Controls properties of existing objects.

**CROSSINGAREACOLOR**

Controls the color of the selection area during crossing selection.

**DRAGMODE**

Controls the way dragged objects are displayed.
HIGHLIGHT
Controls object highlighting; does not affect objects selected with grips.

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

PICKAUTO
Controls automatic windowing at the Select Objects prompt.

PICKBOX
Sets the object selection target height, in pixels.

PICKDRAG
Controls the method of drawing a selection window.

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

PREVIEWEFFECT
Specifies the visual effect used for previewing selection of objects.

PREVIEWFILTER
Excludes specified object types from selection previewing.

SELECTIONAREA
Controls the display of effects for selection areas.

SELECTIONAREAOPACITY
Controls the transparency of the selection area during window and crossing selection.

SELECTIONPREVIEW
Controls the display of selection previewing.

WINDOWAREAECOLOR
Controls the color of the transparent selection area during window selection.
Group Objects

A group is a saved set of objects that you can select and edit together or separately as needed. Groups provide an easy way to combine drawing elements that you need to manipulate as a unit.

See also:
- Work with Blocks on page 405

Overview of Groups

A group is a saved set of objects that you can select and edit together or separately as needed. Groups provide an easy way to combine drawing elements that you need to manipulate as a unit. You can create them quickly and with a default name.

**TIP** Groups are useful in associating 3D solids when you do not want to combine them with a Boolean operation.

You can change the components of groups as you work by adding or removing objects.

In some ways, groups resemble blocks, which provide another method of combining objects into a named set. For example, the groups you create are saved from session to session. However, you can edit individual objects in groups more easily than you can edit them in blocks, which must be exploded first. Unlike blocks, groups cannot be shared with other drawings.

Quick Reference

GROUP
- Creates and manages saved sets of objects called groups.

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

Create Groups

In addition to choosing the objects that will become the members of a group, you can give the group a name and description.
When you create a group, you can give the group a name and description. If you copy a group, the copy is given the default name Ax and is considered unnamed.

If you choose a member of a group that can be selected for inclusion in a new group, all members of the former group are included in the new group.

The objects in your drawing can be members of more than one group, and groups themselves can be nested in other groups. You can ungroup a nested group to restore the original group configuration.

Named groups are not maintained when you use a drawing as an external reference or insert it as a block. However, you can bind and then explode the external reference or explode the block to make the group available as an unnamed group.

**NOTE** Avoid creating large groups containing hundreds or thousands of objects. A large group significantly degrades the performance of this program.

**Quick Reference**

**GROUP**

Creates and manages saved sets of objects called groups.

**PICKSTYLE**

Controls the use of group selection and associative hatch selection.

**Select Objects in Groups**

There are several methods for choosing a group, including selecting the group by name or selecting one of the members of the group.

You can select groups by name at the Select Objects prompt. If the PICKSTYLE system variable is set to 1 or 3 and you select any member of a selectable group, all group members that meet the selection criteria are selected. You can also 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.
Quick Reference

GROUP
Creates and manages saved sets of objects called groups.

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

Edit Groups
You can modify groups in a number of ways, including changing their membership, modifying their properties, revising the names and descriptions of groups, and removing them from the drawing.

Edit Objects as a Group
When group selection is turned on, you can move, copy, rotate, and modify groups just as you can modify individual objects. If you need to edit objects within a group, turn off group selection or use grips to edit individual objects. For more information, see Select Objects in Groups on page 305.

In some circumstances, it is useful to control the order in which objects that belong to the same group are selected. For example, a custom routine that generates toolpaths for numerical control devices might depend on a series of contiguous objects in a specified order.

You can reorder group members in two ways: either change the numerical position of individual members or ranges of group members, or reverse the order of all members. The first object in each group is number 0, not number 1.

Change Group Components, Name, or Description
You can specify objects to be added to or removed from a group at any time. You can also revise a group’s name or description. If deleting an object or removing it from a group leaves the group empty, the group remains defined but without any members.

NOTE Exploding an object such as a block instance or hatch that belongs to a group does not automatically add the resulting components to any group.
Remove Groups
You can delete a group definition by using the “Explode” option. This operation is not the same as exploding a block, hatch, or dimension. Objects that belonged to the exploded group remain in the drawing.
As a result, the group is disbanded but the members are not changed in any other way.

Quick Reference

GROUP
Creates and manages saved sets of objects called groups.

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

Correct Mistakes
You can backtrack your recent actions using one of several methods.

Undo a Single Action
The simplest method of backtracking is to use the UNDO or U commands to undo a single action. Many commands include their own U (undo) option so that you can correct mistakes without leaving the command. When you are creating lines and polylines, for example, enter u to undo the last segment.

NOTE By default, the UNDO command is set to combine consecutive pan and zoom commands into a single operation when you undo or redo. However, pan and zoom commands that are started from the menu are not combined, and always remain separate actions.

Undo Several Actions at Once
Use the Mark option of UNDO to mark an action as you work. You can then use the Back option of UNDO to undo all actions that occurred after the marked action. Use the Begin and End options of UNDO to define a set of actions to be treated as a group.

You can also undo several actions at once with the Undo list on the Standard toolbar.
Reverse the Effect of Undo
You can reverse the effect of a single U or UNDO command by using REDO immediately after using U or UNDO.
You can also redo several actions at once with the Redo list on the Standard toolbar.

Erase Objects
You can erase any object that you draw. If you accidentally erase the wrong object, you can use the UNDO command or the OOPS command to restore it.
For more information, see Erase Objects on page 309.

Cancel a Command
You can cancel a command without completing it by pressing Esc.

Quick Reference
ERASE
   Removes objects from a drawing.
OPTIONS
   Customizes the program settings.
OOPS
   Restores erased objects.
REDO
   Reverses the effects of previous UNDO or U command.
MREDO
   Reverses the effects of several previous UNDO or U commands.
U
   Reverses the most recent operation.
UNDO
   Reverses the effect of commands.
UNDOCTL
Indicates the state of the Auto, Control, and Group options of the UNDO command.

UNDOMARKS
Stores the number of marks placed in the UNDO control stream by the Mark option.

Erase Objects
There are many ways to delete objects from your drawing and clean up the display.

Remove Unused Definitions, Styles, and Objects
You can remove unused named and unnamed objects with PURGE. Some of the unnamed objects you can purge include block definitions, dimension styles, layers, linetypes, and text styles. With PURGE you can also remove zero-length geometry and empty text objects.

Clean Up the Display
You can remove the plus-shaped markers called blips and stray pixels that may be left over from some editing operations from the display area.

- To remove blips, use REDRAW.
- To remove stray pixels, use REGEN.

See also:
- Correct Mistakes on page 307

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

ERASE
Removes objects from a drawing.
OOPS
Restores erased objects.

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

REDRAW
Refreshes the display in the current viewport.

REDRAWALL
Refreshes the display in all viewports.

REGEN
Regenerates the entire drawing from the current viewport.

UNDO
Reverses the effect of commands.

**Cut, Copy, and Paste with the Clipboard**

When you want to use objects from a drawing file in another application, you can cut or copy these objects to the Clipboard and then paste them from the Clipboard into the other application.

**Cut Objects**
Cutting deletes selected objects from the drawing and stores them on the Clipboard. The objects are now available to be pasted into other programs.

**Copy Objects**
You can use the Clipboard to copy part or all of a drawing into a document created by another application. The objects are copied in vector format, which retains the high resolution in other applications. The information stored in the Clipboard can then be pasted in other programs.

**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.
**Quick Reference**

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

**COPYCLIP**
Copies selected objects to the Clipboard.

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

**PASTECLIP**
Pastes objects from the Clipboard into the current drawing.

---

**Modify Objects**

You can modify the size, shape, and location of objects.

**See also:**
- [Work with Custom and Proxy Objects](#) on page 937
- [Modify Existing Dimensions](#) on page 819

---

**Choose a Method to Modify Objects**

Access object editing options using the following methods:

<table>
<thead>
<tr>
<th>Methods</th>
<th>Descriptions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Command line</td>
<td>Enter a command and then select the objects to modify. Alternatively, select the objects first and then enter a command.</td>
</tr>
<tr>
<td>Shortcut menu</td>
<td>Select and right-click an object to display a shortcut menu with relevant editing options.</td>
</tr>
<tr>
<td>Double-click</td>
<td>Double-click an object to display the Properties Inspector palette or, in some cases, a dialog box or editor that is specific to that type of object.</td>
</tr>
</tbody>
</table>
### Descriptions

Control grip behavior with the following methods:

- **Grip Modes.** Click a grip and right-click to select one of the Grip modes (Stretch, Move, Rotate, Scale, or Mirror).

- **Multi-functional grip-editing options.** Select a polyline, spline, or non-associative polyline hatch object and hover over a grip to access additional options for reshaping the object.

### See also:

- [Change Text on page 749](#)
- [Select Objects on page 293](#)
- [Modify Existing Dimensions on page 819](#)
- [Display and Change the Properties of Objects on page 163](#)
- [Work with Custom and Proxy Objects on page 937](#)

### Quick Reference

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DRAGMODE</td>
<td>Controls the way dragged objects are displayed.</td>
</tr>
<tr>
<td>PROPERTIES</td>
<td>Controls properties of existing objects.</td>
</tr>
<tr>
<td>SELECT</td>
<td>Places selected objects in the Previous selection set.</td>
</tr>
<tr>
<td>DBLCLKEDIT</td>
<td>Controls the double click editing behavior in the drawing area.</td>
</tr>
</tbody>
</table>
DRAGMODE
Controls the way dragged objects are displayed.

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

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

Edit Objects with Grips
Grips are displayed at strategic points on selected objects. Manipulate grips to modify objects.

Use Grip Modes
Grip modes control how an object behaves when manipulated with grips. Use Grip modes to stretch, move, scale, rotate, or mirror an object.

To use a Grip mode, select a grip (base grip) to act as the base point for the action. (A selected grip is also called a hot grip.) Then press Enter or Spacebar to cycle through the Grip modes. You can also use shortcut keys or right-click to see all of the modes.

NOTE Grips are not displayed on objects that are on locked layers.
Stretch with Grips
You can stretch an object by moving selected grips to new locations. Grips on text, block references, midpoints of lines, centers of circles, and point objects move the object rather than stretching it. This is an excellent method for moving block references and adjusting dimensions.

NOTE When a 2D object lies on a plane other than the current UCS, the object is stretched on the plane on which it was created, not on the plane of the current UCS.

Move with Grips
You can move objects by the grip selected. Selected objects are highlighted and are moved the direction and distance of the next point location you specify.

Scale with Grips
You can scale selected objects relative to a base point. Increase the size of an object by dragging outward from the base grip and specifying a point location, or decrease the size by dragging inward. Alternatively, you can enter a value for relative scaling.

Mirror with Grips
You can mirror selected objects across a temporary mirror line. Turning Ortho mode on helps you specify a vertical or horizontal mirror line.

Rotate with Grips
You can rotate selected objects around a base point by dragging and specifying a point location. Alternatively, you can enter an angle value. This is an excellent method for rotating block references.

Select and Modify Multiple Grips
You can use more than one grip as the base grips for the action. When you select more than one grip (also called multiple hot grip selection), the shape of the object is kept intact between the selected grips. To select more than one grip, press and hold the Shift key, and then select the appropriate grips.

Limit the Display of Grips to Improve Performance
You can limit the maximum number of objects that display grips. For example, when a drawing contains hatch objects or polylines with many grips, selecting...
these objects can take a long time. The GRIPOBJLIMIT system variable suppresses the display of grips when the initial selection set includes more than the specified number of objects. If you add objects to the current selection set, the limit does not apply.

**NOTE** Grips are not displayed on objects that are on locked layers.

### Work with Quadrant Grips

For quadrant grips on circles and ellipses, distance is measured from the center point, not the selected grip. For example, in Stretch mode, you can select a quadrant grip to stretch a circle and then specify a distance at the Command prompt for the new radius. The distance is measured from the center of the circle, not the selected quadrant. If you select the center point to stretch the circle, the circle moves.

**See also:**
- Choose a Method to Modify Objects on page 311
- Modify Polylines on page 355
- Modify Hatches and Fills on page 697
- Use Dynamic Input on page 223

### Quick Reference

**OPTIONS**

- Customizes the program settings.

**GRIPBLOCK**

- Controls the display of grips in blocks.

**GRIPCOLOR**

- Controls the color of unselected grips.

**GRIPCOUTURE**

- Controls the color of the grip contour.

**GRIPHOT**

- Controls the color of selected grips.
GRIPHOVER
Controls the fill color of an unselected grip when the cursor pauses over it.

GRIPOBJLIMIT
Suppresses the display of grips when the selection set includes more than the specified number of objects.

GRIPS
Controls the display of grips on selected objects.

GRIPSIZE
Sets the size of the grip box in pixels.

GRIPTIPS
Controls the display of grip tips and Ctrl-cycling tooltips.

**Modify Objects with Multi-Functional Grips**
Modify polylines, splines, and non-associative polyline hatch objects with multi-functional grips.

Control the display of these grips with the GRIPS system variable.

With multi-functional grips, you can

- **Modify the position, size, and orientation of objects.** Use **Grip modes** on page 313 to move, rotate, scale, or mirror objects.
- **Reshape Objects.** Use the multi-functional grip-editing options to edit vertices, fit points, control points, segment types, and tangent directions.

To activate a multi-functional grip, select the grip, or hover over it and choose a multi-functional grip-editing option from the dynamic menu. Once a grip is active, change the grip behavior with the Hot Grip shortcut menu, or by cycling through options as follows:

- Press Enter or Spacebar to cycle through the Grip modes
- Press Ctrl to cycle through the multi-functional grip-editing options.

**NOTE** If the object is constrained, the first Ctrl relaxes constraints.
Control the access methods to multi-functional grips with the GRIPMULTIFUNCTIONAL system variable.

**Quick Reference**

**OPTIONS**
- Customizes the program settings.

**GRIPBLOCK**
- Controls the display of grips in blocks.

**GRIPCOLOR**
- Controls the color of unselected grips.

**GRIPCONTOUR**
- Controls the color of the grip contour.

**GRIPHOT**
- Controls the color of selected grips.

**GRIPHOVER**
- Controls the fill color of an unselected grip when the cursor pauses over it.

**GRIPOBJLIMIT**
- Suppresses the display of grips when the selection set includes more than the specified number of objects.

**GRIPS**
- Controls the display of grips on selected objects.

**GRIPSIZE**
- Sets the size of the grip box in pixels.

**GRIPTIPS**
- Controls the display of grip tips and Ctrl-cycling tooltips.

**GRIPMULTIFUNCTIONAL**
- Specifies the access methods to multi-functional grips.
**Make Multiple Copies with Grips**

You can create multiple copies of objects as you modify them with any of the grip modes.

For example, by using the Copy option, you can rotate the selected objects, leaving copies at each location you specify with the pointing device.

![Multiple Copies Diagram](image)

You can also make multiple copies by holding down Ctrl as you select the first point. For example, with the Stretch grip mode, you can stretch an object, such as a line, and then copy it to any point in the drawing area. Multiple copies continue being made until you turn off grips.

**NOTE** When you use grips to make multiple copies of an object that contains multiple, only the current scale representation is copied.

**Define an Offset Snap or a Rotation Snap**

You can place multiple copies at regularly spaced intervals with an offset snap. The offset snap is defined by the distance between an object and the next copy. In the lighting layout below, the first copy of the light fixture symbol is placed at an offset of two units. All subsequent copies are then placed two units apart.

![Offset Snap Diagram](image)

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.

**Quick Reference**

**OPTIONS**
Customizes the program settings.

**GRIPBLOCK**
Controls the display of grips in blocks.

**GRIPCOLOR**
Controls the color of unselected grips.

**GRIPCONTOUR**
Controls the color of the grip contour.

**GRIPHOT**
Controls the color of selected grips.
GRIPS
Controls the display of grips on selected objects.

GRIPSIZE
Sets the size of the grip box in pixels.

Control Grips in Blocks
You can specify whether a block displays a single grip or multiple grips.
You can specify whether a selected block reference displays a single grip at its insertion point or displays multiple grips associated with the objects grouped within the block.

See also:
■ Use Grip Modes on page 313

Quick Reference
OPTIONS
Customizes the program settings.
GRIPBLOCK
Controls the display of grips in blocks.
GRIPCOLOR
Controls the color of unselected grips.
GRIPCOUTOUR
Controls the color of the grip contour.
GRIPHOT

Controls the color of selected grips.

GRIPS

Controls the display of grips on selected objects.

GRIPS SIZE

Sets the size of the grip box in pixels.

Move or Rotate Objects

You can move objects to a different location, or change the orientation of objects by rotating them by an angle or to other objects.

Move Objects

You can move objects at a specified distance and direction from the originals.

Use coordinates, grid snap, object snaps, and other tools to move objects with precision.

Specify Distance with Two Points

Move an object using the distance and direction specified by a base point followed by a second point. In this example, you move the block representing a window. Select the object to be moved (1). Specify the base point for the move (2) followed by a second point (3). The object is moved the distance and direction of point 2 to point 3.

Use a Stretch-Move

You can also use STRETCH to move objects if all their endpoints lie entirely within the selection window. Turn on Ortho mode or polar tracking to move the objects at a specific angle.
A practical example is moving a door in a wall. The door in the illustration is entirely within a crossing selection, while the wall lines are only partly within the crossing selection area.

The result is that only the endpoints that lie within the crossing selection move.

See also:
- Edit Objects with Grips on page 313

Quick Reference

CHSPACE
Moves objects between model space and paper space.

MOVE
Moves objects a specified distance in a specified direction.

PROPERTIES
Controls properties of existing objects.

STRETCH
Stretches objects crossed by a selection window or polygon.

SNAPUNIT
Sets the snap spacing for the current viewport.

Direct Distance Entry (Command Modifier)
Locates the next point at a specified distance in the direction of your cursor.

Rotate Objects
You can rotate objects in your drawing around a specified base point.
To determine the angle of rotation, you can enter an angle value, drag using the cursor, or specify a reference angle to align to an absolute angle.

**Rotate an Object by a Specified Angle**

Enter a rotation angle value from 0 to 360 degrees. You can also enter values in radians, grads, or surveyor bearings. Entering a positive angle value rotates the objects counterclockwise or clockwise, depending on the base angle direction setting in the Drawing Units dialog box.

**Rotate an Object by Dragging**

Drag the object around the base point and specify a second point. Use Ortho mode, polar tracking, or object snaps for greater precision.

For example, you can rotate the plan view of a house by selecting the objects (1), specifying a base point (2), and specifying an angle of rotation by dragging to another point (3).

**Rotate an Object to an Absolute Angle**

With the Reference option, you can rotate an object to align it to an absolute angle.

For example, to rotate the part in the illustration so the diagonal edge rotates to 90 degrees, you select the objects to be rotated (1, 2), specify the base point (3), and enter the Reference option. For the reference angle, specify the two endpoints of the diagonal line (4, 5). For the new angle, enter 90.
Rotate an Object in 3D

To rotate 3D objects, you can use either ROTATE or ROTATE3D.

- With ROTATE, you can rotate objects around a specified base point. The axis of rotation passes through the base point and is parallel to the Z axis of the current UCS.
- With ROTATE3D, you can specify the axis of rotation using either two points; an object; the X, Y, or Z axis; or the Z direction of the current view.

See also:
- Rotate Views in Layout Viewports on page 155

Quick Reference

ROTATE
   Rotates objects around a base point.

ROTATE3D
   Moves objects about a 3D axis.

Align Objects

You can move, rotate, or tilt an object so that it aligns with another object.

In the following example, two pairs of points are used to align the piping in 2D using the ALIGN command. Endpoint object snaps align the pipes precisely.
In 3D, use the 3DALIGN command to specify up to three points to define the source plane followed by up to three points to define the destination plane.

- The first source point on an object, called the base point, is always moved to the first destination point.
- Specifying a second point for either the source or the destination results in the selected objects being rotated.
- A third point for either the source or the destination results in further rotation of the selected objects.

**TIP** With 3D solid models, it is recommended that you turn on dynamic UCS to speed the selection of the destination plane.

### Quick Reference

**3DALIGN**
Aligns objects with other objects in 2D and 3D.

**ALIGN**
Aligns objects with other objects in 2D and 3D.

**UCSDTECT**
Controls whether dynamic UCS acquisition is active or not.

### Copy, Offset, or Mirror Objects
You can create duplicates of objects in your drawing that are either identical or similar to selected objects.
Copy Objects

See also:

- Edit Objects with Grips on page 313
- Enter Direct Distances on page 247
- Create an Array of Objects on page 327

You can create duplicates of objects at a specified distance and direction from the originals.

Use coordinates, grid snap, object snaps, and other tools to copy objects with precision.

You can also use grips to move and copy objects quickly.

Specify Distance with Two Points

Copy an object using the distance and direction specified by a base point followed by a second point. In this example, you copy the block representing an electronic component. Select the original object to be copied. Specify the base point for the move (1) followed by a second point (2). The object is copied the distance and direction of point 1 to point 2.

Specify Distance with Relative Coordinates

Copy an object using a relative distance by entering coordinate values for the first point and pressing Enter for the second point. The coordinate values are used as a relative displacement rather than the location of a base point.

NOTE Do not include an @ sign as you normally would for relative coordinates, because relative coordinates are expected.
To copy objects a specified distance, you can also use direct distance entry with Ortho mode and polar tracking.

**Create Multiple Copies**

By default, COPY prompts you to create multiple copies from the specified selection set and base point. Position copies at specified displacements, or arrange a set number of copies in a linear array.

---

**Quick Reference**

**ADDSELECTED**

Creates a new object based on the object type and general properties of a selected object.

**COPY**

Copies objects a specified distance in a specified direction.

**COPYMODE**

Controls whether the COPY command repeats automatically.

---

**Create an Array of Objects**

You can create copies of objects in a rectangular or polar (circular) pattern called an array.

For rectangular arrays, you control the number of rows and columns and the distance between each. For polar arrays, you control the number of copies of the object and whether the copies are rotated. To create many regularly spaced objects, arraying is faster than copying.
Create Rectangular Arrays

A rectangular array is built along a baseline defined by the current snap rotation angle. This angle is zero by default, so the rows and columns of a rectangular array are orthogonal with respect to the $X$ and $Y$ axes. The default angle 0 direction setting can be changed in UNITS.

Create Polar Arrays

When you create a polar array, the array is drawn counterclockwise or clockwise, depending on whether you enter a positive or a negative value for the angle to fill.

The radius of the array is determined by the distance from the specified center point to a reference or base point on the last selected object. You can use the default reference point (usually an arbitrary point that coincides with a snap point), or you can specify a new base point to be used as the reference point.
Array in 3D

With 3DARRAY, you can create a rectangular array or a polar array of objects in 3D. In addition to specifying the number of columns (X direction) and rows (Y direction), you also specify the number of levels (Z direction).

Limit the Size of Arrays

If you specify a very large number of rows and columns for an array, it may take a long time to create the copies. By default, the number of array elements that can be generated by one command is limited to approximately 100,000. This limit is controlled by the MaxArray setting in the registry.

You can change the limit by setting the MaxArray system registry variable using (`setenv "MaxArray" "n"`) where `n` is a number between 100 and 1000000 (ten million).

NOTE When changing the value of MaxArray, you must enter MaxArray with the capitalization shown.

To create a rectangular array

1. Click Home tab ➤ Modify panel ➤ Array.
2. In the Array dialog box, select Rectangular Array.
3. Click Select Objects.
   The Array dialog box closes. You are prompted for object selection.
4. Select the objects to be arrayed and press Enter.
5. In the Rows and Columns boxes, enter the number of rows and columns in the array.
6. Specify the horizontal and vertical spacing (offsets) between objects by using one of the following methods:
   - In the Row Offset and Column Offset boxes, enter the distance between rows and between columns. Adding a plus sign (+) or a minus sign (-) determines direction.
   - Click the Pick Both Offsets button to use the pointing device to specify the diagonal corners of a cell in the array. The cell determines the vertical and horizontal spacing of the rows and columns.
Click the Pick Row Offset or Pick Column Offset button to use the pointing device to specify the horizontal and vertical spacing.

The example box displays the result.

7 To change the rotation angle of the array, enter the new angle next to Angle of Array.

8 The default angle 0 direction setting can also be changed in UNITS.

9 Click OK to create the array.

To create a polar array

1 Click Home tab ➤ Modify panel ➤ Array.

2 In the Array dialog box, select Polar Array.

3 Next to Center Point, do one of the following:
   ■ Enter an X value and a Y value for the center point of the polar array.
   ■ Click the Pick Center Point button. The Array dialog box closes and you are prompted for object selection. Use the pointing device to specify the center point of the polar array.

4 Click Select Objects.
   The Array dialog box closes and you are prompted for object selection.

5 Select the objects to be arrayed.

6 In the Method box, select one of the following methods:
   ■ Total Number of Items & Angle to Fill
   ■ Total Number of Items & Angle Between Items
   ■ Angle to Fill & Angle Between Items

7 Enter the number of items (including the original object), if available.

8 Use one of the following methods:
   ■ Enter the angle to fill and angle between items, if available. Angle to Fill specifies the distance to fill around the circumference of the array. Angle Between Items specifies the distance between each item.
Click the Pick Angle to Fill button and the Pick Angle Between Items button. Use the pointing device to specify the angle to fill and the angle between items.

The example box displays the result.

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.

Click OK to create the array.

**To create a 3D rectangular array of objects**

1. Click Home tab ➤ Modify panel ➤ 3D Array.
2. Select the object to array (1).
3. Specify Rectangular.
4. Enter the number of rows.
5. Enter the number of columns.
6. Enter the number of levels.
7. Specify the distance between rows.
8. Specify the distance between columns.
9. Specify the distance between levels.

Copy, Offset, or Mirror Objects | 331
To create a 3D polar array of objects

1. Click Home tab ➤ Modify panel ➤ 3D Array.
2. Select the object to array (1).
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).

Quick Reference

3DARRAY

Creates a 3D matrix of objects in a rectangular or polar arrangement.
ARRAY

Creates multiple copies of objects in a pattern.

DSETTINGS

Sets grid and snap, polar and object snap tracking, object snap modes, and Dynamic Input.

UCS

Manages user coordinate systems.

UNITS

Controls coordinate and angle display formats and precision.

GETENV

Shows values of specified system registry variables.

SETENV

Sets values of specified registry variables.

ANGBASE

Sets the base angle to 0 with respect to the current UCS.

ANGDIR

Sets the direction of positive angles.

SNAPANG

Sets the snap and grid rotation angle for the current viewport relative to the current UCS.

Offset an Object

Offset an object to create a new object whose shape parallels the shape of the original object.

OFFSET creates a new object whose shape parallels the shape of a selected object. Offsetting a circle or an arc creates a larger or smaller circle or arc, depending on which side you specify for the offset.
A highly effective drawing technique is to offset objects and then trim or extend their ends.

You can offset
- Lines
- Arcs
- Circles
- Ellipses and elliptical arcs (resulting in an oval-shaped spline)
- 2D polylines
- Construction lines (xlines) and rays
- Splines

**Special Cases for Offset Polylines and Splines**

2D polylines are offset as individual line segments, resulting in either intersections or gaps between segments. To complete the offset, intersecting lines are trimmed and gaps are filled.

Splines are trimmed automatically when the offset distance is larger than can otherwise be accommodated.
Quick Reference

OFFSET
Creates concentric circles, parallel lines, and parallel curves.
OFFSETDIST
Sets the default offset distance.
OFFSETGAPTYPE
Controls how potential gaps between segments are treated when polylines are offset.

Mirror Objects

You can flip objects about a specified axis to create a symmetrical mirror image. Mirroring is useful for creating symmetrical objects because you can quickly draw half the object and then mirror it instead of drawing the entire object.

You flip objects about an axis called a mirror line to create a mirror image. To specify this temporary mirror line, you enter two points. You can choose whether to erase or retain the original objects.

By default, when you mirror text, hatches, attributes, and attribute definitions, they are not reversed or turned upside down in the mirror image. The text has the same alignment and justification as before the object was mirrored. If you do want text to be reversed, set the MIRRTEXT system variable to 1.
MIRRTEXT affects text that is created with the TEXT, ATTDEF, or MTEXT commands; attribute definitions; and variable attributes. Text and constant attributes that are part of an inserted block are reversed when the block is mirrored regardless of the value of MIRRTEXT.

MIRRTEXT affects hatch objects created with the GRADIENT or HATCH commands. Use the MIRRHATCH system variable control whether hatch pattern direction is mirrored or retained.

**Mirror in 3D**

With MIRROR3D, you can mirror objects across a specified mirroring plane. The mirroring plane can be one of the following:

- The plane of a planar object
- A plane parallel to the XY, YZ, or XZ plane of the current UCS that passes through a specified point
- A plane defined by three specified points (2, 3, and 4)

**Quick Reference**

**MIRROR**

Creates a mirrored copy of selected objects.

**MIRROR3D**

Creates a mirrored copy of selected objects across a mirroring plane.
MIRRATCH
Controls how MIRROR reflects hatch patterns.

MIRRTXT
Controls how MIRROR reflects text.

Change the Size and Shape of Objects
There are several methods for adjusting the lengths of existing objects relative to other objects, both symmetrically and asymmetrically.

Trim or Extend Objects
You can shorten or lengthen objects to meet the edges of other objects.
This means you can first create an object such as a line and then later adjust it to fit exactly between other objects.
Objects you select as cutting edges or boundary edges are not required to intersect the object being trimmed. You can trim or extend an object to a projected edge or to an extrapolated intersection; that is, where the objects would intersect if they were extended.
If you do not specify a boundary and press Enter at the Select Objects prompt, all displayed objects become potential boundaries.

NOTE To select cutting edges or boundary edges that include blocks, you can use only the single selection, Crossing, Fence, and Select All options.

Trim Objects
You can trim objects so that they end precisely at boundary edges defined by other objects.
For example, you can clean up the intersection of two walls smoothly by trimming.
An object can be one of the cutting edges and one of the objects being trimmed. For example, in the illustrated light fixture, the circle is a cutting edge for the construction lines and is also being trimmed.

When you trim several objects, the different selection methods can help you choose the current cutting edges and objects to trim. In the following example, the cutting edges are selected using crossing selection.

The following example uses the fence selection method to select a series of objects for trimming.
You can trim objects to their nearest intersection with other objects. Instead of selecting cutting edges, you press Enter. Then, when you select the objects to trim, the nearest displayed objects act as cutting edges. In this example, the walls are trimmed so that they intersect smoothly.

You can extend objects without leaving the TRIM command. Press and hold 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.

**NOTE** You can trim objects without leaving the EXTEND command. Press and hold 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 351

Quick Reference

BREAK

Breaks the selected object between two points.

EXTEND

Extends objects to meet the edges of other objects.
JOIN
Joins the endpoints of lines, 2D and 3D polylines, arcs, elliptical arcs, helixes, and splines to create single object.

LENGTHEN
Changes the length of objects and the included angle of arcs.

PROPERTIES
Controls properties of existing objects.

TRIM
Trims objects to meet the edges of other objects.

EDGEMODE
Controls how the TRIM and EXTEND commands determine cutting and boundary edges.

PROJMODE
Sets the current Projection mode for trimming or extending.

Resize or Reshape Objects
You can resize objects to make them longer or shorter in only one direction or to make them proportionally larger or smaller.

You can also stretch certain objects by moving an endpoint, vertex, or control point.

Lengthen Objects
With LENGTHEN, you can change the included angle of arcs and the length of the following objects:
- Lines
- Arcs
- Open polylines
- Elliptical arcs
- Open splines.
The results are similar to extending and trimming. You can

- Drag an object endpoint dynamically
- Specify a new length or angle as a percentage of the total length or angle
- Specify an incremental length or angle measured from an endpoint
- Specify the object’s total absolute length or included angle

**Stretch Objects**

With STRETCH, you relocate the endpoints of objects that lie across or within a crossing selection window.

- Objects that are partially enclosed by a crossing window are stretched.
- Objects that are completely enclosed within the crossing window, or that are selected individually, are moved rather than stretched.

You stretch an object by specifying a base point and then a point of displacement.

Use object snaps, grid snaps, and relative coordinate entry to stretch with precision.

**Scale Objects Using a Scale Factor**

With SCALE, you can make an object uniformly larger or smaller. To scale an object, you specify a base point and a scale factor. Alternatively, you can specify a length to be used as a scale factor based on the current drawing units.

A scale factor greater than 1 enlarges the object. A scale factor between 0 and 1 shrinks the object.

Scaling changes the size of all dimensions of the selected object. A scale factor greater than 1 enlarges the object. A scale factor less than 1 shrinks the object.
NOTE When you use the SCALE command with objects, the position or location of the object is scaled relative to the base point of the scale operation, but the size of the object is not changed.

Scale Objects Using a Reference Distance

You can also scale by reference. Scaling by reference uses an existing distance as a basis for the new size. To scale by reference, specify the current distance and then the new desired size. For example, if one side of an object is 4.8 units long and you want to expand it to 7.5 units, use 4.8 as the reference length.

You can use the Reference option to scale an entire drawing. For example, use this option when the original drawing units need to be changed. Select all objects in the drawing. Then use Reference to select two points and specify the intended distance. All the objects in the drawing are scaled accordingly.

See also:

■ Break and Join Objects on page 351

Quick Reference

JOIN

Joins the endpoints of lines, 2D and 3D polylines, arcs, elliptical arcs, helixes, and splines to create single object.

LENGTHEN

Changes the length of objects and the included angle of arcs.

PEDIT

Edits polylines and 3D polygon meshes.

PROPERTIES

Controls properties of existing objects.
SCALE
Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling.

SPLINEDIT
Modifies the parameters of a spline or converts a spline-fit polyline to a spline.

STRETCH
Stretches objects crossed by a selection window or polygon.

PLINECONVERTMODE
Specifies the fit method used in converting splines to polylines.

Fillet, Chamfer, Break, or Join Objects
You can change objects to meet in rounded or flattened corners. You can also create or close gaps in objects.

Create Fillets
A fillet connects two objects with an arc that is tangent to the objects and has a specified radius.

An inside corner is called a fillet and an outside corner is called a round; you can create both using the FILLET command.

You can fillet
- Arcs
- Circles
- Ellipses and elliptical arcs
- Lines
- Polylines
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.

![Diagram showing filleted objects with different radii](image)

You can press and hold Shift while selecting the objects to override the current fillet radius with a value of 0.

**Trim and Extend Filleted Objects**

You can use the Trim option to specify whether the selected objects are trimmed or extended to the endpoints of the resulting arc or left unchanged.
Control the Location of the Fillet

Depending on the locations you specify, more than one possible fillet can exist between the selected objects. Compare the selection locations and resulting fillets in the illustrations.

Fillet Line and Polyline Combinations

To fillet lines with polylines, each line or its extension must intersect one of the polyline line segments. If the Trim option is on, the filleted objects and the fillet arc join to form a single new polyline.

Fillet an Entire Polyline

You can fillet an entire polyline or remove fillets from an entire polyline. If you set a nonzero fillet radius, FILLET inserts fillet arcs at the vertex of each polyline segment that is long enough to accommodate the fillet radius.
If two polyline line segments converge as they approach an arc segment that separates them, FILLET removes the arc segment and replaces it with a fillet arc.

If you set the fillet radius to 0, no fillet arcs are inserted. If two polyline line segments are separated by one arc segment, FILLET removes that arc and extends the lines until they intersect.

**Fillet Parallel Lines**

You can fillet parallel lines, xlines, and rays. The current fillet radius temporarily adjusts to create an arc that is tangent to both objects and located in the plane common to both objects.

The first selected object must be a line or a ray, but the second object can be a line, an xline, or a ray. The fillet arc connects as shown in the illustration.

**Fillet Objects with Non-Zero Thickness in 3D**

You can fillet coplanar objects with extrusion directions not parallel to the Z axis of the current UCS. FILLET determines the extrusion direction for the fillet arc in 3D space closest to the direction of the Z axis of the current UCS.
Quick Reference

FILLET
Rounds and fillets the edges of objects.

FILLETRAD
Stores the current fillet radius for 2D objects.

TRIMMODE
Controls whether selected edges for chamfers and fillets are trimmed.

Create Chamfers

A chamfer connects two objects to meet in a flattened or beveled corner.
A chamfer connects two objects with an angled line. It is usually used to represent a beveled edge on a corner.

You can chamfer

■ Lines
■ Polylines
■ Rays
■ Xlines
■ 3D solids

CHAMFER can be used to bevel all corners of a polyline using a single command.

NOTE Chamfering a hatch boundary that was defined from line segments removes hatch associativity. If the hatch boundary was defined from a polyline, associativity is maintained.

If both objects being chamfered are on the same layer, the chamfer line is created on that layer. Otherwise, the chamfer line is created on the current layer. The layer affects object properties including color and linetype.
Use the Multiple option to chamfer more than one set of objects without leaving the command.

**Chamfer by Specifying Distances**

The chamfer distance is the amount each object is trimmed or extended to meet the chamfer line or to intersect the other. If both chamfer distances are 0, chamfering trims or extends the two objects until they intersect but does not create a chamfer line. You can press and hold Shift while selecting the objects to override the current chamfer distances with a value of 0.

In the following example, you set the chamfer distance to 0.5 for the first line and 0.25 for the second line. After you specify the chamfer distance, you select the two lines as shown.

**Trim and Extend Chamfered Objects**

By default, objects are trimmed when chamfered, but you can use the Trim option to specify that they remain untrimmed.

**Chamfer by Specify Length and Angle**

You can chamfer two objects by specifying where on the first selected object the chamfer line starts, and then the angle the chamfer line forms with this object.

In this example, you chamfer two lines so that the chamfer line starts 1.5 units from the intersection along the first line and forms an angle of 30 degrees with this line.
Chamfer Polylines and Polyline Segments

If the two objects you select for chamfering are segments of a polyline, they must be adjacent or separated by no more than one arc segment. If they are separated by an arc segment, as shown in the illustration, chamfering deletes the arc and replaces it with a chamfer line.

Chamfer an Entire Polyline

When you chamfer an entire polyline, each intersection is chamfered. For best results, keep the first and second chamfer distances equal.

In this example, the chamfer distances are set to equal values.

When you chamfer an entire polyline, only the segments that are long enough to accommodate the chamfer distance are chamfered. The polyline in the following illustration has some segments too short to be chamfered.
Quick Reference

CHAMFER
Bevels the edges of objects.

CHAMFERA
Sets the first chamfer distance when CHAMMODE is set to 0.

CHAMFERB
Sets the second chamfer distance when CHAMMODE is set to 0.

CHAMFERC
Sets the chamfer length when CHAMMODE is set to 1.

CHAMFERD
Sets the chamfer angle when CHAMMODE is set to 1.

CHAMMODE
Sets the input method for CHAMFER

PREVIEWCREATIONTRANSPARENCY

TRIMMODE
Controls whether selected edges for chamfers and fillets are trimmed.

Break and Join Objects

You can break an object into two objects with or without a gap between them.
You can also join objects to create single object or multiple objects.

Break Objects
Use BREAK to create a gap in an object, resulting in two objects with a gap
between them. BREAK is often used to create space for block or text.
To break an object without creating a gap, specify both break points at the same location.

You can create breaks in most geometric objects except blocks, dimensions, multilines, and regions. As an alternative, use EXPLODE on these types of objects, and create breaks in the dissociated geometry.

**Join Objects**

The result of the join operation varies depending on the objects selected. Typical applications include

- Replacing two collinear lines with a single line.
- Closing the gap in a line that resulted from a BREAK.
- Completing an arc into a circle or an elliptical arc into an ellipse. To access the Close option, select a single arc or elliptical arc.
- Combining several long polylines in a topographic map.
- Joining two splines, leaving a kink between them.

In general cases, joining objects that touch end-to-end, but that are not in the same plane result in 3D polylines and splines.

**NOTE** You can also use the Join option of the PEDIT command to combine a series of lines, arcs, and polylines into a single polyline

**See also:**

- **Modify Polylines** on page 355
- **Modify Complex Objects** on page 353
Quick Reference

BREAK
Breaks the selected object between two points.

JOIN
Joins the endpoints of lines, 2D and 3D polylines, arcs, elliptical arcs, helixes, and splines to create single object.

Modify Complex Objects

Additional editing operations are available for complex objects, such as blocks, dimensions, hatches, and polylines.

See also:
- Create 3D Models on page 441
- Modify Blocks on page 431
- “Modify Hatches and Solid-Filled Areas” on page 23
- Change Text on page 749
- Modify Existing Dimensions on page 819
- Update Fields on page 735
- Create and Modify Tables on page 757
- Modify Raster Images and Image Boundaries on page 913

Disassociate Compound Objects (Explode)

You can convert a compound object, such as a polyline, dimension, hatch, or block reference, into individual elements.

You can explode a compound object, such as a polyline, dimension, hatch, or block reference, to convert it into individual elements. For example, exploding a polyline breaks it down to simple lines and arcs. Exploding a block reference or an associative dimension replaces it with copies of the objects that compose the block or dimension.
Explode Dimensions and Hatches

When you explode a dimension or a hatch, all associativity is lost and the dimension or hatch object is replaced by individual objects such as lines, text, points, and 2D solids. To explode dimensions automatically when you create them, set the DIMASSOC system variable to 0.

Explode Polylines

When you explode a polyline, any associated width information is discarded. The resulting lines and arcs follow the polyline’s centerline. If you explode a block that contains a polyline, you need to explode the polyline separately. If you explode a donut, its width becomes 0.

Explode Block References

If you explode a block with attributes, the attribute values are lost, leaving only the attribute definitions. The colors and linetypes of objects in exploded block references can change.

NOTE Blocks inserted with MINsert (multiple insert) result in an minsert block object, and cannot be exploded directly. You can convert the minsert block object into a block object with the FLATTEN Express Tool.

Explode External References

An external reference (xref) is a drawing file linked (or attached) to another drawing. You cannot explode xrefs and their dependent blocks.

Quick Reference

EXPLODE

Breaks a compound object into its component objects.

XPLODE

Breaks a compound object into its component objects.

DIMASSOC

Controls the associativity of dimension objects and whether dimensions are exploded.
EXPLMODE

Controls whether the EXPLODE command supports nonuniformly scaled (NUS) blocks.

Modify Polyline

Change the shape and display of polyline objects with polyline editing options. You can also join separate polylines.

For example, modify polylines in several ways using PEDIT, the Properties Inspector palette, or grips.

- Move, add, or delete individual vertices
- Set a uniform width for the entire polyline or control the width of each segment
- Create an approximation of a spline called a spline-fit polyline
- Display noncontinuous linetypes with or without a dash before and after each vertex
- Change the orientation of text in a polyline's linetype by reversing its direction

Modify Polyline with Multi-Functional Grips

Polyline grips are multi-functional, providing context-sensitive options for reshaping the polyline. Polyline segments have additional multi-functional grips displayed at their midpoints. Control the display of these grips with the GRIPS system variable.

NOTE  Multi-functional grips also support the standard Grip modes (Stretch, Move, Rotate, Scale, and Mirror). For information on working with multi-functional grips, see Modify Objects with Multi-Functional Grips on page 316.

The multi-functional grip-editing options that are available for polylines depend on

- The grip's location (vertex or midpoint)
- The segment type (line or arc)
The type of Polyline (standard, curve-fit, or spline-fit)

**Polyline multi-functional grip editing options**

<table>
<thead>
<tr>
<th>Option</th>
<th>Animation</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Stretch or Stretch Vertex</strong></td>
<td>Specify a stretch point.</td>
</tr>
<tr>
<td>Add Vertex</td>
<td>Specify a point for the new vertex.</td>
</tr>
<tr>
<td>Remove Vertex</td>
<td>Delete the selected vertex.</td>
</tr>
<tr>
<td>Convert to Arc</td>
<td>Specify the midpoint of a straight segment to convert it to an arc segment.</td>
</tr>
<tr>
<td>Convert to Line</td>
<td>Specify the midpoint of an arc segment to convert into a straight segment.</td>
</tr>
<tr>
<td>Tangent Direction</td>
<td>Manipulate the tangent directions to redefine the shape of a curve-fit polyline.</td>
</tr>
</tbody>
</table>

**Modify a Segment Within a Polyline**

To select individual segments (or subobjects) of a polyline, press Ctrl while clicking the segments. Use Grip modes and multi-functional grip-editing options to modify the polyline segments. The GRIPSUBOBJMODE system variable controls whether grips are automatically made hot when subobjects are selected.

![multi-functional grips displayed on selected subobjects](image)
You can modify the width of individual segments with the Properties Inspector palette. Changing any other property for a polyline segment affects the entire polyline.

**Modify Polylines with Coincident Grips**

Coincident grips are grips that are shared between multiple objects. When polyline objects sharing the same grip are selected together, multi-functional grip-editing options are not supported for the coincident grip. However, you can still edit the coincident grip using the standard Grip modes (Stretch, Move, Rotate, Scale, and Mirror).

**NOTE** Multi-functional grip-editing options for coincident grips are supported when only one of the polyline objects is selected.

**Join Polyline Segments**

You can join a line, an arc, or another polyline to an open polyline if their ends connect or are close to each other.

If the ends are not coincident but are within a distance that you can set, called the *fuzz distance*, the ends are joined by either trimming them, extending them, or connecting them with a new segment.

Spline-fit polylines return to their original shape when joined. Polylines cannot be joined into a Y shape.

If the properties of several objects being joined into a polyline differ, the resulting polyline inherits the properties of the first object that you selected.

See also:
- Choose a Method to Modify Objects on page 311
- Use Grip Modes on page 313
- Modify Objects with Multi-Functional Grips on page 316
- Overview of Constraints on page 367
- Trim or Extend Objects on page 337
- Break and Join Objects on page 351
Quick Reference

PEDIT
Edits polylines and 3D polygon meshes.

JOIN
Joins the endpoints of lines, 2D and 3D polylines, arcs, elliptical arcs, helixes, and splines to create a single object.

REVERSE
Reverses the vertices of selected lines, polylines, splines, and helixes, which is useful for linetypes with included text, or wide polylines with differing beginning and ending widths.

GRIPS
Controls the color of selected grips.

GRIPMULTIFUNCTIONAL
Specifies the access methods to multi-functional grips.

GRIPSUBOBJMODE
Controls whether grips are automatically made hot when subobjects are selected.

PEDITACCEPT
Suppresses display of the Object Selected Is Not a Polyline prompt in PEDIT.

PLINECONVERTMODE
Specifies the fit method used in converting splines to polylines.

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

SPLINETYPE
Sets the type of curve generated by the Spline option of the PEDIT command.

SURFTYPE
Controls the type of surface-fitting to be performed by the Smooth option of the PEDIT command.
SURFU
Sets the surface density for PEDIT Smooth in the M direction and the U isolines density on surface objects.

SURFV
Sets the surface density for PEDIT Smooth in the N direction and the V isolines density on surface objects.

Modify Splines

See also:
- **Draw Splines** on page 281
- **Break and Join Objects** on page 351
- **Modify Objects with Multi-Functional Grips** on page 316
- **Edit NURBS Surfaces** on page 600
- **Rebuild NURBS Surfaces and Curves** on page 601

Several methods are available for editing splines and changing their underlying mathematical parameters.

Spline editing can be accomplished with multi-functional grips, with SPLINEDIT, and with the Properties Inspector palette.

- Multi-functional grip options include adding control vertices and changing the tangent direction of the spline at its endpoints.

- The Properties Inspector palette provides access to several spline parameters and options, including the degree of the spline, the weight of each control point, and whether the spline is closed.
- SPLINEDIT provides additional editing options such as adding a kink to the spline, and joining a spline to another object such as a line, arc, or other spline located at an endpoint of the original spline (C0 continuity).

- Trimming a spline shortens it without changing the shape of the portion that remains.

**Switching Between Control Vertices and Fit Points**

The editing options available with multi-functional grips differ depending whether the spline is set to display control vertices (CVs) or fit points. The spline on the left displays control vertices, and the one on the right displays fit points.

To switch between displaying control vertices and displaying fit points, click the triangular grip.

**IMPORTANT** Switching from displaying control vertices to fit points automatically changes the selected spline to degree 3. Splines originally created using higher-degree equations will likely change shape as a result.

360 | Chapter 18  Change Existing Objects
Editing Splines With Control Vertices

In general, editing a spline with control vertices provides finer control over reshaping a small section of the curve than editing a spline with fit points.

You can insert additional control vertices to a section of a spline to obtain greater control in that section at the expense of making the shape of the spline more complicated.

The Refine option adds a knot to the spline resulting in replacing the selected control vertex with two control vertices.

For more information, see Modify Objects with Multi-Functional Grips on page 316.

NOTE Because periodic curves and surfaces are not currently supported, the objects may kink if they are reshaped.
Quick Reference

Commands

CVSHOW
Displays the control vertices for specified NURBS surfaces or curves.

CVADD
Adds control vertices to NURBS surfaces and splines.

CVHIDE
Turns off the display of control vertices for all NURBS surfaces and curves.

CVREBUILD
Rebuilds the shape of NURBS surfaces and curves.

CVREMOVE
Removes control vertices from NURBS surfaces and curves.

JOIN
Joins the endpoints of lines, 2D and 3D polylines, arcs, elliptical arcs, helixes, and splines to create single object.

SPLINE
Creates a smooth B-spline curve that passes through or near a set of points that controls the shape of the curve.

SPLINEDIT
Modifies the parameters of a spline or converts a spline-fit polyline to a spline.

REVERSE
Reverses the vertices of selected lines, polylines, splines, and helixes, which is useful for linetypes with included text, or wide polylines with differing beginning and ending widths.

System Variables

GRIPS
Controls the display of grips on selected objects.
PLINECONVERTMODE

Specifies the fit method used in converting splines to polylines.

Modify Helixes

You can use grips or the Properties Inspector palette to modify the shape and size of a helix.

You can use the grips on a helix to change the following properties:

- Start point
- Base radius
- Top radius
- Height
- Location

When you use a grip to change the base radius of a helix, the top radius scales to maintain the current ratio. Use the Properties Inspector palette to change the base radius independent of the top radius.

You can use the Properties Inspector palette to change other helix properties, such as

- Number of turns (Turns)
- Turn height
- Direction of the twist — clockwise (CW) or counterclockwise (CCW)

With the Constrain property, you can specify that the Height, Turns, or Turn Height properties of the helix are constrained. The Constrain property affects how the helix changes when the Height, Turns, or Turn Height properties are changed either through the Properties Inspector palette or through grip editing. The table below shows the behavior of the helix depending on which property is constrained.

<table>
<thead>
<tr>
<th>Constrained property</th>
<th>Property to change</th>
<th>Effect on these helix properties</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Height</td>
<td>Height</td>
</tr>
<tr>
<td>Height</td>
<td></td>
<td>Changed</td>
</tr>
<tr>
<td></td>
<td>Turns</td>
<td>Fixed</td>
</tr>
<tr>
<td></td>
<td>Turn Height</td>
<td>Changed</td>
</tr>
</tbody>
</table>
### Effect on these helix properties

<table>
<thead>
<tr>
<th>Constrained</th>
<th>Property to change</th>
<th>Effect on these helix properties</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Height</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Fixed</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Fixed</td>
</tr>
<tr>
<td>Turn Height</td>
<td>Height</td>
<td>Changed</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Fixed</td>
</tr>
<tr>
<td>Turn Height</td>
<td>Changing</td>
<td>Changed</td>
</tr>
<tr>
<td></td>
<td>Changing</td>
<td>Changed</td>
</tr>
<tr>
<td></td>
<td>Fixed</td>
<td>Changed</td>
</tr>
</tbody>
</table>

**See also:**
- [Draw Helixes](#) on page 285

### Quick Reference

**HELIX**

Creates a 2D spiral or 3D spring.

### Modify Multilines

Multiline objects are composed of 1 to 16 parallel lines, called elements. To modify multilines or their elements, you can use common multiline editing commands.

Special multiline editing features are available with the MEDIT command including the following:
- Add or delete a vertex
- Control the visibility of corner joints
Control the style of intersection with other multilines

Open or close gaps in a multiline object

**Add and Delete Multiline Vertices**

You can add or delete any vertex in a multiline.

```
| vertex in multiline to delete | multiline with vertex deleted |
```

**Edit Multiline Intersections**

If you have two multilines in a drawing, you can control the way they intersect. Multilines can intersect in a cross or a T shape, and the crosses or T shapes can be closed, open, or merged.

```
| closed cross | open T | merged cross |
```

**Use Common Editing Commands on Multilines**

You can use most of the common editing commands on multilines *except*

- **BREAK**
- **CHAMFER**
- **FILLET**
- **LENGTHEN**
- **OFFSET**

To perform these operations, first use EXPLODE to replace the multiline object with separate line objects.
NOTE If you trim or extend a multiline object, only the first boundary object encountered determines the shape of the end of the multiline. A multiline cannot have a complex boundary at its endpoint.

See also:
- Draw Multiline Objects on page 267

**Quick Reference**

**MLEDIT**

Edits multiline intersections, breaks, and vertices.
Add Constraints to Geometry

With parametric drawing, you can add constraints to geometry to ensure that the design conforms to specified requirements.

Overview of Constraints

Parametric drawing is a technology that is used for designing with constraints. Constraints are associations and restrictions applied to 2D geometry.

There are two general types of constraints:

- Geometric constraints control the relationships of objects with respect to each other
- Dimensional constraints control the distance, length, angle, and radius values of objects

The following illustration displays geometric and dimensional constraints using the default format and visibility.

A blue cursor icon always displays when you move the cursor over an object that has constraints applied to it.
In the design phase of a project, constraints provide a way to enforce requirements when experimenting with different designs or when making changes. Changes made to objects can adjust other objects automatically, and restrict changes to distance and angle values.

With constraints, you can

■ Maintain design specifications and requirements by constraining the geometry within a drawing
■ Apply multiple geometric constraints to objects instantly
■ Include formulas and equations within dimensional constraints
■ Make design changes quickly by changing the value of a variable

**BEST PRACTICE** It is recommended that you first apply geometric constraints to determine the *shape* of a design, and then apply dimensional constraints to determine the *size* of objects in a design.

**Design Using Constraints**

When you are creating or changing a design, a drawing will be in one of three states:

■ *Unconstrained*. No constraints are applied to any geometry.
■ *Underconstrained*. Some constraints are applied to the geometry.
■ *Fully constrained*. All relevant geometric and dimensional constraints are applied to the geometry. A fully constrained set of objects also needs to include at least one Fix constraint to lock the location of the geometry.
Thus, there are two general methods for designing with constraints:

- You can work in an underconstrained drawing and make changes as you go, using a combination of editing commands, grips, and adding or changing constraints.
- You can create and fully constrain a drawing first, and then control the design exclusively by relaxing and replacing geometric constraints, and changing the values in dimensional constraints.

The method that you choose depends on your design practices and the requirements of your discipline.

**NOTE** The program prevents you from applying any constraints that result in an overconstrained condition.

### Remove or Relax Constraints

There are two ways to cancel the effects of constraints when you need to make design changes:

- Delete the constraints individually and later apply new constraints. While the cursor hovers over a geometric constraint icon, you can use the Delete key or the shortcut menu to delete the constraint.
- Relax the constraints temporarily on selected objects to make the changes. With a grip selected or when you specify options during an editing command, tap the Ctrl key to alternate between relaxing constraints and maintaining constraints.

Relaxed constraints are not maintained during editing. Constraints are restored automatically if possible when the editing process is complete. Constraints that are no longer valid are removed.

**NOTE** The DELCONSTRAINT command deletes all geometric and dimensional constraints from an object.

### Quick Reference

**AUTOCONSTRAIN**

Applies geometric constraints to a selection set of objects based on orientation of the objects relative to one another.
CONSTRAINTBAR
Displays or hides the geometric constraints on an object.

CONSTRAINTSETTINGS
Controls the display of geometric constraints on constraint bars.

DCDISPLAY
Displays or hides the dynamic constraints associated with a selection set of objects.

DELCONSTRAINT
Removes all geometric and dimensional constraints from a selection set of objects.

DIMCONSTRAINT
Applies dimensional constraints to selected objects or points on objects, or converts associative dimensions to dimensional constraints.

GEOMCONSTRAINT
Displays or hides the geometric constraints on an object.

LIST
Displays property data for selected objects.

PARAMETERS (-PARAMETERS)
Controls the associative parameters used in the drawing.

TEXTEDIT
Edits a dimensional constraint, dimension, or text object.

CCONSTRAINTFORM
Controls whether annotational or dynamic constraints are applied to objects.

CONSTRAINTBARDISPLAY
Controls the display of constraint bars after you apply constraints and when you select geometrically constrained drawings.

CONSTRAINTBARMODE
Controls the display of geometrical constraints on constraint bars.

CONSTRAINTNAMEFORMAT
Controls the text format for dimensional constraints.
CONSTRANTRELAX
Indicates whether constraints are enforced or relaxed when editing an object.

CONSTRAINTSOLVEMODE
Controls constraint behavior when applying or editing constraints.

DIMCONSTRAINTICON
Displays the lock icon next to the text for dimensional constraints.

DYNCONSTRAINTMODE
Displays hidden dimensional constraints when constrained objects are selected.

PARAMETERCOPYMODE
Controls how constraints and referenced user parameters are handled when constrained objects are copied between drawings, Model space and layouts, and block definitions.

Constrain Objects Geometrically
Geometric constraints determine the relationships between 2D geometric objects or points on objects relative to each other.

Overview of Geometric Constraints
You can specify geometric constraints between 2D objects or points on objects. When you later edit the constrained geometry, the constraints are maintained.

Thus, using geometric constraints, you have a method of including design requirements in your drawing.

For example, in the illustration below, the following constraints are applied to the geometry.

- Every endpoint is constrained to remain coincident with the endpoint of every adjacent object—these constraints are displayed as small blue squares
- The vertical lines are constrained to remain parallel with each other and to remain equal to each other in length
- The left vertical line is constrained to remain perpendicular to the horizontal line
The horizontal line is constrained to remain horizontal.

The location of the circle and the horizontal line are constrained to remain fixed in space—these constraints are displayed as lock icons.

**NOTE** The locked geometry is not associated to the other geometry without geometric constraints linked to it.

The geometry is not *fully constrained*, however. Using grips, you can still change the radius of the arc, the diameter of the circle, the length of the horizontal line, and the length of the vertical lines. To specify these distances, you need to apply dimensional constraints.

**NOTE** Constraints can be added to segments within a polyline as if they were separate objects.

**See also:**
- Overview of Dimensional Constraints on page 386

**Quick Reference**

**AUTOCONSTRAIN**

Applies geometric constraints to a selection set of objects based on orientation of the objects relative to one another.

**CONSTRAINTBAR**

Displays or hides the geometric constraints on an object.

**CONSTRAINTSETTINGS**

Controls the display of geometric constraints on constraint bars.
DELCONSTRAINT
Removes all geometric and dimensional constraints from a selection set of objects.

GEOMCONSTRAINT
Displays or hides the geometric constraints on an object.

LIST
Displays property data for selected objects.

CONSTRAINTBARMODE
Controls the display of geometrical constraints on constraint bars.

CONSTRAINTBARDISPLAY
Controls the display of constraint bars after you apply constraints and when you select geometrically constrained drawings.

CONSTRAINTNAMEFORMAT
Controls the text format for dimensional constraints.

CONSTRAINTRELAX
Indicates whether constraints are enforced or relaxed when editing an object.

CONSTRAINTSOLVEMODE
Controls constraint behavior when applying or editing constraints.

**Apply or Remove Geometric Constraints**
Geometric constraints associate geometric objects together, or specify a fixed location or angle.

For example, you can specify that a line should always be perpendicular to another one, that an arc and a circle should always remain concentric, or that a line should always be tangent to an arc.
When you apply a constraint, two things occur:

- The object that you select adjusts automatically to conform to the specified constraint.
- By default, a gray constraint icon displays near the constrained object as shown in the previous illustration, and a small blue glyph displays with your cursor when you move it over a constrained object.

Once applied, constraints permit only those changes to the geometry that do not violate the constraints. This provides a method for exploring design options or making design changes while maintaining the requirements and specifications of the design.

**NOTE** The order in which you select two objects when you apply a constraint is important in some cases. Normally, the second object you select adjusts to the first object. For example, when you apply a perpendicular constraint, the second object you select will adjust to become perpendicular to the first.

You can apply geometric constraints to 2D geometric objects only. Objects cannot be constrained between model space and paper space.
Specify Constraint Points

With some constraints, you specify *constraint points* on objects instead of selecting the objects. This behavior is similar to that of object snaps, but the locations are limited to endpoints, midpoints, center points, and insertion points.

For example, a coincident constraint can restrict the location of the endpoint of one line to the endpoint of another line.

The following glyph is displayed on the object as you roll over the object.

![Glyph Image]

You use this glyph to confirm whether you are specifying the intended point to constrain.

The fix, horizontal, and vertical constraint icons indicate whether the constraints are applied to an object or a point.

<table>
<thead>
<tr>
<th>Constraint</th>
<th>Point</th>
<th>Object</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fix</td>
<td>![Fix Icon]</td>
<td>![Fix Icon]</td>
</tr>
<tr>
<td>Horizontal</td>
<td>![Horizontal Icon]</td>
<td>![Horizontal Icon]</td>
</tr>
<tr>
<td>Vertical</td>
<td>![Vertical Icon]</td>
<td>![Vertical Icon]</td>
</tr>
</tbody>
</table>

The symmetric constraint icons indicate whether it identifying a symmetrical point or object, or the symmetrical line.

<table>
<thead>
<tr>
<th>Constraint</th>
<th>Point</th>
<th>Object</th>
<th>Line</th>
</tr>
</thead>
<tbody>
<tr>
<td>Symmetric</td>
<td>![Symmetric Icon]</td>
<td>![Symmetric Icon]</td>
<td>![Symmetric Icon]</td>
</tr>
</tbody>
</table>

When rolling over any icon, the constraint point markers are displayed indicating the constrained points. You do not need to roll over the icon to identify the constraints that are applied to the points of the selected object.
A different set of constraint bar icons are displayed when a horizontal or vertical constraint is not parallel or perpendicular with the current UCS.

Use Fix Constraints

A fix constraint associates a constraint point on an object, or the object itself with a fixed location with respect to the World Coordinate System.

It is often advisable to specify a fix constraint at an important geometric feature. This locks the location of that point or object, and prevents geometry from relocating when you make changes to the design.

When you fix an object, the angle of a line, or the center of an arc or circle is also fixed.

Apply Multiple Geometric Constraints

You can apply multiple geometric constraints to objects either manually or automatically.

When you want to apply all essential geometric constraints to a design automatically, you can use AUTOCONSTRAIN with the objects that you select in your drawing. This helps constrain the geometric shape of the design—depending on your design, there might be cases where you need to apply additional geometric constraints.

AUTOCONSTRAIN also provides settings in which you can specify the following options:

- What geometric constraints to apply
- What order to apply geometric constraints
- What tolerances are used to determine whether objects are horizontal, vertical, or touching
**NOTE** Fix constraint is not applied with AUTOCONSTRAIN. You must apply the constraint individually. Equal constraint applied with AUTOCONSTRAIN resizes the selected arcs to the same radius only. It is not applied to the arc length.

To fully constrain the size and proportions of a design, you will later need to apply dimensional constraints.

**Remove Geometric Constraints**

A geometric constraint cannot be modified, but you can delete it and apply a different one. Several constraint options, including Delete, are available from the shortcut menu that is displayed when you right-click a constraint icon in the drawing.

You can delete all constraints from a selection set in a single operation with DELCONSTRAINT.

**Quick Reference**

**CONSTRAINTBAR**

Displays or hides the geometric constraints on an object.

**CONSTRAINTSETTINGS**

Controls the display of geometric constraints on constraint bars.

**GCCINCIDENT**

Constrains two points together or a point to a curve (or an extension of a curve).

**GCCOLLINEAR**

Causes two or more line segments to lie along the same line.

**GCCONCENTRIC**

Constrains two arcs, circles, or ellipses to the same center point.

**GCEQUAL**

Resizes selected arcs and circles to the same radius, or selected lines to the same length.

**GCFIX**

Locks points and curves in position.
GCHORIZONTAL
Causes lines or pairs of points to lie parallel to the X axis of the current coordinate system.

GCPARALLEL
Causes selected lines to lie parallel to each other.

GCPERPENDICULAR
Causes selected lines to lie 90 degrees to one another.

GCsmooth
Constrains a spline to be contiguous and maintain G2 continuity with another spline, line, arc, or polyline.

GCSYMMETRIC
Causes selected objects to become symmetrically constrained about a selected line.

GCTANGENT
Constrains two curves to maintain a point of tangency to each other or their extensions.

GCVERTICAL
Causes lines or pairs of points to lie parallel to the Y axis of the current coordinate system.

GEOMCONSTRAINT
Displays or hides the geometric constraints on an object.

CONSTRAINTBARMODE
Controls the display of geometrical constraints on constraint bars.

CONSTRAINTBARDISPLAY
Controls the display of constraint bars after you apply constraints and when you select geometrically constrained drawings.

CONSTRAINTNAMEFORMAT
Controls the text format for dimensional constraints.

CONSTRAINTRELAX
Indicates whether constraints are enforced or relaxed when editing an object.
**CONSTRAINTSOLVEMODE**

Controls constraint behavior when applying or editing constraints.

**Display and Verify Geometric Constraints**

You can determine visually what objects are associated with any geometric constraint, or what constraints are associated with any object.

*Constraint icons* provide information about how objects are constrained. A *constraint bar* displays one or more icons that represent the geometric constraints applied to an object.

You can drag constraint bars when you need to move them out of the way, and you can also control whether they are displayed or hidden.

**Verify the Geometric Constraints on Objects**

You can confirm the association of geometric constraints with objects in two ways.

- When you roll over a *constraint icon* on a constraint bar, the objects associated with that geometric constraint are highlighted.

- When you roll over an *object* that has geometric constraints applied to it, all constraint bars that are associated with the object are highlighted.
These highlighting features simplify working with constraints especially when you have many constraints applied throughout a drawing.

**Control the Display of Constraint Bars**

Geometric constraints and constraint bars can be displayed or hidden, either individually or globally. You can do any of the following:

- Display or hide all geometric constraints
- Display or hide specified types of geometric constraints
- Display or hide all geometric constraints associated with a selected object
- Temporarily display the geometric constraints of the selected object

Use the Constraint Settings dialog box to control the types of geometric constraints that are displayed or hidden on constraint bars.

You can set the constraint bars to automatically and temporarily display when the constrained geometry is selected. When the geometry is no longer selected, the temporarily displayed constraint bars are hidden.

Hiding geometric constraints is useful when you analyze a design and want to filter the display of geometric constraints. For example, you can choose to display the icons for Parallel constraints only. Next, you might choose to display the icons for Perpendicular constraints only.
NOTE To reduce clutter, Coincident constraints display by default as small, light-blue squares. You can use an option in the Constraint Settings dialog box to turn them off if necessary.

Quick Reference

GEOMCONSTRAINT
Displays or hides the geometric constraints on an object.

CONSTRAINTBAR
Displays or hides the geometric constraints on an object.

CONSTRAINTSETTINGS
Controls the display of geometric constraints on constraint bars.

CONSTRAINTBARMODE
Controls the display of geometrical constraints on constraint bars.

CONSTRAINTBARDISPLAY
Controls the display of constraint bars after you apply constraints and when you select geometrically constrained drawings.

CONSTRAINTNAMEFORMAT
Controls the text format for dimensional constraints.

CONSTRAINTRELAX
Indicates whether constraints are enforced or relaxed when editing an object.

CONSTRAINTSOLVEMODE
Controls constraint behavior when applying or editing constraints.

Modify Objects with Geometric Constraints Applied

You can edit constrained geometric objects with grips, editing commands, or by relaxing or applying geometric constraints.

By definition, geometric constraints that are applied to geometric objects limit the editing actions that you perform on the objects.
**Modify Constrained Objects with Grips**

You can modify constrained geometry using grip editing modes. The geometry will maintain all applied constraints.

For example, if a line object is constrained to remain tangent to a circle, you can rotate the line and change its length and endpoints, but the line or its extension will remain tangent to the circle.

If the circle was an arc instead, the line or its extension would remain tangent to the arc or its extension.

The results of modifying underconstrained objects are based on what constraints have already been applied and the object types involved. For example, if the Radius constraint had not been applied, the radius of the circle would have been modified instead of the tangent point of the line.

The CONSTRAINTSOLVEMODE system variable determines the way an object behaves when constraints are applied or when grips are used to edit it.

**BEST PRACTICE** You can limit unexpected changes by applying additional geometric or dimensional constraints. Common choices include coincident and fix constraints.
Modify Constrained Objects with Editing Commands

You can use editing commands such as MOVE, COPY, ROTATE, SCALE, and STRETCH to modify constrained geometry. The results maintain the constraints applied to the objects.

**NOTE** The TRIM, EXTEND, BREAK, and JOIN commands in some circumstances can remove constraints.

By default, if an editing command results in copying the constrained objects, the constraints applied to the original objects will also be duplicated. This behavior is controlled by the PARAMETERCOPYMODE system variable. Using the copying technique, you can save work by taking advantage of multiple instances of objects, bilateral symmetry, or radial symmetry.

For information about temporarily relaxing constraints, see Overview of Constraints on page 367.

Quick Reference

**GEOMCONSTRAINT**
- Displays or hides the geometric constraints on an object.

**CONSTRAINTBAR**
- Displays or hides the geometric constraints on an object.

**CONSTRAINTSETTINGS**
- Controls the display of geometric constraints on constraint bars.

**CONSTRAINTBARMODE**
- Controls the display of geometrical constraints on constraint bars.

**CONSTRAINTBARDISPLAY**
- Controls the display of constraint bars after you apply constraints and when you select geometrically constrained drawings.

**CONSTRAINTNAMEFORMAT**
- Controls the text format for dimensional constraints.

**CONSTRAINTRELAX**
- Indicates whether constraints are enforced or relaxed when editing an object.
CONSTRAINTSOLVEMODE

Controls constraint behavior when applying or editing constraints.

**Infer Geometric Constraints**

You can automatically apply geometric constraints while creating and editing geometric objects.

Enabling *Infer Constraints* mode automatically applies constraints between the object you are creating or editing and the object or points associated with object snaps.

Similar to the AUTOCONSTRAIN command, constraints are applied only if the objects meet the constraint conditions. Objects are not repositioned as a result of inferring constraints.

With Infer Constraints turned on, the object snaps that you specify when creating geometry are used to infer geometric constraints. However, the following object snaps are not supported: Intersection, Apparent Intersection, Extension, and Quadrant.

The following constraints cannot be inferred:

- Fix
- Smooth
- Symmetric
- Concentric
- Equal
- Collinear

**Infer Constraints with Line and Polyline**

Certain object creation and editing commands can infer constraints regardless of the current object snap settings.

LINE and PLINE commands infer coincident *point-to-point* constraints. The Close option infers a coincident constraint between the start point of the first line and the endpoint of the last line.
Infer Constraints with Rectangle, Fillet, and Chamfer

The RECTANG, FILLET, and CHAMFER commands infer constraints as follows:

- RECTANG applies a pair of parallel constraints and a perpendicular constraint to the closed polyline.
- FILLET applies tangent and coincident constraints between the newly created arc and the existing trimmed or extended pair of lines.
- CHAMFER applies coincident constraints between the newly created line and the existing trimmed or extended pair of lines.

The following commands are unaffected by the Infer Constraints setting:

- SCALE
- MIRROR
- OFFSET
- BREAK
- TRIM
- EXTEND
- ARRAY

Infer Constraints with Move, Copy, and Stretch

When moving, copying, or stretching with the Infer Constraints on, you can apply coincident, perpendicular, parallel, or tangent constraints between the object being edited and the object being snapped to if the base point of the edited object is a valid constraint point of that object.

For example, if a line is stretched and snapped to an endpoint of another line, a coincident constraint is applied between the endpoints of the two lines.

A vertical or horizontal constraint can be applied between objects when you move, copy, or stretch an object from a valid constraint point while object tracking vertically or horizontally along a valid constraint point on another object.

See also:

- Use Object Snaps on page 226
Quick Reference

CHAMFER
- Bevels the edges of objects.

CONSTRAINTSETTINGS
- Controls the display of geometric constraints on constraint bars.

DIMCONSTRAINT
- Applies dimensional constraints to selected objects or points on objects, or converts associative dimensions to dimensional constraints.

FILLET
- Rounds and fillets the edges of objects.

LINE
- Creates straight line segments.

PLINE
- Creates a 2D polyline.

RECTANG
- Creates a rectangular polyline.

CONSTRAINTINFER
- Controls whether the geometric constraints are inferred while drawing and editing geometry.

Constrain Distances and Angles between Objects

You can control distances or angles between 2D geometric objects or points on objects applying dimensional constraints and specifying values. You can also constrain geometry with variables and equations.

Overview of Dimensional Constraints

Dimensional constraints control the size and proportions of a design. They can constrain the following:

- Distances between objects, or between points on objects
- Angles between objects, or between points on objects
- Sizes of arcs and circles

For example, the following illustration includes linear, aligned, angular, and diameter constraints.

If you change the value of a dimensional constraint, all the constraints on the object are evaluated, and the objects that are affected are updated automatically.

Also, constraints can be added directly to segments within a polyline as if they were separate objects.

**NOTE** The number of decimal places displayed in dimensional constraints is controlled by the LUPREC and AUPREC system variables.

**Compare Dimensional Constraints with Dimension Objects**

Dimensional constraints are different from dimension objects in the following ways:

- Dimensional constraints are used in the design phase of a drawing, but dimensions are typically created in the documentation phase
- Dimensional constraints drive the size or angle of objects, but dimensions are driven by objects
- By default, dimensional constraints are not objects, display with only a single dimension style, maintain the same size during zoom operations, and are not outputted to a device
If you need to output a drawing with dimensional constraints or use dimension styles, you can change the form of a dimensional constraint from dynamic to annotational. See Apply Dimensional Constraints on page 389 for more detail.

**Define Variables and Equations**

The -PARAMETERS command allows you to define custom *user variables* that you can reference from within dimensional constraints and other user variables. The expressions that you define can include a variety of predefined functions and constants.

For more information about using variables and equations with constraints, see Constrain a Design with Formulas and Equations on page 398

**See also:**
- Overview of Geometric Constraints on page 371
- Apply Dimensional Constraints on page 389
- Constrain a Design with Formulas and Equations on page 398

**Quick Reference**

**DCDISPLAY**
- Displays or hides the dynamic constraints associated with a selection set of objects.

**DELCONSTRAINT**
- Removes all geometric and dimensional constraints from a selection set of objects.

**DIMCONSTRAINT**
- Applies dimensional constraints to selected objects or points on objects, or converts associative dimensions to dimensional constraints.

**LIST**
- Displays property data for selected objects.

**PARAMETERS (-PARAMETERS)**
- Controls the associative parameters used in the drawing.
Apply Dimensional Constraints

Dimensional constraints maintain specified distances and angles between geometric objects or points on objects.

For example, you can specify that the length of a line should always remain at 6.00 units, that the vertical distance between two points be maintained at 1.00 unit, and that a circle should always remain at 1.00 unit in diameter.
When you apply a dimensional constraint to an object, a *constraint variable* is automatically created for maintaining the constraint value. By default, these are assigned names such as *d1* or *dia1*, but you can rename them with the -PARAMTERS.

Dimensional constraints can be created in one of the following forms:

- Dynamic constraints
- Annotational constraints

The forms have different purposes. In addition, any dynamic or annotational constraint can be converted to a *reference parameter*.

**Dynamic Constraints**

By default, dimensional constraints are *dynamic*. They are ideal for normal parametric drawing and design tasks.

Dynamic constraints have the following characteristics:

- Maintain the same size when zooming in or out
- Can easily be turned on or off globally in the drawing
- Display using a fixed, predefined dimension style
- Position the textual information automatically, and provide triangle grips with which you can change the value of a dimensional constraint
- Do not display when the drawing is plotted

If you need to control the dimension style of dynamic constraints, or if you need to plot dimensional constraints, use the Properties Inspector to change dynamic constraints to annotational constraints.
**Annotational Constraints**

Annotational constraints are useful when you want dimensional constraints to have the following characteristics:

- Change their size when zooming in or out
- Display individually with layers
- Display using the *current* dimension style
- Provide grip capabilities that are similar to those on dimensions
- Display when the drawing is plotted

**NOTE** To display the text used in annotational constraints in the same format as used in dimensions, set the CONSTRAINTNAMEFORMAT system variable to 1.

After plotting, you can use the Properties Inspector to convert annotational constraints back to dynamic constraints.

**Reference Parameters**

A reference parameter is a *driven* dimensional constraint, either dynamic or annotational. This means that it does not control the associated geometry, but rather reports a measurement similar to a dimension object.

You use reference parameters as a convenient way to display measurements that you would otherwise have to calculate. For example, the width in the illustration is constrained by the diameter constraint, \( d_{1} \), and the linear constraint, \( d_{1} \). The reference parameter, \( d_{2} \), displays the total width but does not constrain it. The textual information in reference parameters is always displayed within parentheses.
You can set the Reference property in the Properties Inspector to convert a dynamic or annotational constraint to a reference parameter.

**NOTE** You cannot change a reference parameter back to a dimensional constraint if doing so would overconstrain the geometry.

## Quick Reference

**DCALIGNED**
Constrains the distance between two points on different objects.

**DCANGULAR**
Constrains the angle between line or polyline segments, the angle swept out by an arc or a polyline arc segment, or the angle between three points on objects.

**DCCONVERT**
Converts associative dimensions to dimensional constraints.

**DCDIAMETER**
Constrains the diameter of a circle or an arc.

**DCDISPLAY**
Displays or hides the dynamic constraints associated with a selection set of objects.

**DCFORM**
Specifies whether the dimensional constraint being created is dynamic or annotational.

**DCHORIZONTAL**
Constrains the X distance between points on an object, or between two points on different objects.

**DCLINEAR**
Creates a horizontal, vertical, or rotated constraint based on the locations of the extension line origins and the dimension line.

**DCRADIUS**
Constrains the radius of a circle or an arc.
DCVERTICAL
Constrains the Y distance between points on an object, or between two points on different objects.

DELCONSTRAINT
Removes all geometric and dimensional constraints from a selection set of objects.

DIMCONSTRAINT
Applies dimensional constraints to selected objects or points on objects, or converts associative dimensions to dimensional constraints.

LIST
Displays property data for selected objects.

PARAMETERS (-PARAMETERS)
Controls the associative parameters used in the drawing.

TEXTEDIT
Edits a dimensional constraint, dimension, or text object.

CONSTRAINTNAMEFORMAT
Controls the text format for dimensional constraints.

CONTRAINTRELAX
Indicates whether constraints are enforced or relaxed when editing an object.

CONSTRAINTSOLVEMODE
Controls constraint behavior when applying or editing constraints.

DIMCONSTRAINTICON
Displays the lock icon next to the text for dimensional constraints.

DYNCONSTRAINTMODE
Displays hidden dimensional constraints when constrained objects are selected.

PARAMETERCOPYMODE
Controls how constraints and referenced user parameters are handled when constrained objects are copied between drawings, Model space and layouts, and block definitions.
Control the Display of Dimensional Constraints

You can display or hide dynamic and annotational constraints within a drawing.

Display or Hide Dynamic Constraints

You can hide all dynamic constraints to reduce clutter when you want to work with geometric constraints only, or when you need to continue other work in the drawing. You can turn on their display when needed from the ribbon or with the DCDISPLAY command.

By default, if you select an object associated with a hidden dynamic constraint, all dynamic constraints associated with that object are temporarily displayed.

You can display or hide the dynamic constraints for all objects or for a selection set.

Display or Hide Annotational Constraints

You control the display of annotational constraints as you would with dimension objects—you assign them to a layer and turn the layer on or off as needed. You can also specify object properties for annotational constraints such as dimension style, color, and lineweight.

Quick Reference

DCDISPLAY

Displays or hides the dynamic constraints associated with a selection set of objects.

DELCONSTRAINT

Removes all geometric and dimensional constraints from a selection set of objects.

DIMCONSTRAINT

Applies dimensional constraints to selected objects or points on objects, or converts associative dimensions to dimensional constraints.

LIST

Displays property data for selected objects.

PARAMETERS (-PARAMETERS)

Controls the associative parameters used in the drawing.
Modify Objects with Dimensional Constraints Applied

You can control lengths, distances, and angles of objects by changing constraint values, by manipulating dimensional constraints using grips, or by changing user variables or expressions associated with dimensional constraints.

Edit Dimensional Constraint Names, Values, and Expressions

You can edit the names, values, and expressions that are associated with dimensional constraints using in-place editing:

- Double-click the dimensional constraint, select the dimensional constraint and use the shortcut menu, or the TEXTEDIT command
- Open the Properties Inspector and select the dimensional constraint

You can reference other dimensional constraints by selecting them during an in-place editing operation.
NOTE You cannot edit the Expression and Value properties for a reference parameter.

Modify Dimensional Constraints Using Their Grips

You can modify a constrained object either by using the triangular grips or the square grips on the associated dimensional constraint.

The triangular grips on dimensional constraints provide a way of changing the constraint value while maintaining the constraint.

For example, you can change the length of the diagonal line by using the triangular grips on the Aligned dimensional constraint. The diagonal line maintains its angle and the location of one of its endpoints.

The square grip on dimensional constraints provides a way of changing the location of the text and other elements.

Dynamic dimensional constraints are more limited than annotational dimensional constraints in where the text can be located.

NOTE Triangular grips are not available for dimensional constraints that reference other constraint variables in expressions.
For information about temporarily relaxing constraints, see Overview of Constraints on page 367.

See also:
- Overview of Constraints on page 367

Quick Reference

DCDISPLAY
Displays or hides the dynamic constraints associated with a selection set of objects.

DELCONSTRAINT
Removes all geometric and dimensional constraints from a selection set of objects.

DIMCONSTRAINT
Applies dimensional constraints to selected objects or points on objects, or converts associative dimensions to dimensional constraints.

LIST
Displays property data for selected objects.

PARAMETERS (-PARAMETERS)
Controls the associative parameters used in the drawing.

TEXTEDIT
Edits a dimensional constraint, dimension, or text object.

CONSTRAINTNAMEFORMAT
Controls the text format for dimensional constraints.

CONSTRAINTRELAX
Indicates whether constraints are enforced or relaxed when editing an object.

CONSTRAINTSOLVEMODE
Controls constraint behavior when applying or editing constraints.

DIMCONSTRAINTICON
Displays the lock icon next to the text for dimensional constraints.
DYNCONSTRAINTMODE
Displays hidden dimensional constraints when constrained objects are selected.

PARAMETERCOPYMODE
Controls how constraints and referenced user parameters are handled when constrained objects are copied between drawings, Model space and layouts, and block definitions.

Constrain a Design with Formulas and Equations
You can control geometry using mathematical expressions that include the names of dimensional constraints, user variables, and functions.

Overview of Formulas and Equations
Formulas and equations can be represented either as expressions within dimensional constraint parameters or by defining user variables. For example, the following illustration represents a design that constrains a circle to the center of the rectangle with an area equal to that of the rectangle.

The Length and Width dimensional constraint parameters are set to constants. The d1 and d2 constraints are simple expressions that reference the Length and Width. The Radius dimensional constraint parameter is set to an expression that includes the square root function, parentheses to determine the precedence of operations, the Area user variable, the division operator, and the constant, PI.

As you can see, part of the equation for determining the area of the circle is included in the Radius dimensional constraint parameter and part was defined as a user variable. Alternatively, the entire expression, sqrt (Length * Width /
PI), could have been assigned to the Radius dimensional constraint parameter, defined in a user variable, or some other combination.

**Protect Expressions in Dynamic Constraints**

When a dynamic dimensional constraint references one or more parameters, the prefix `fx:` is added to the name of the constraint. This prefix is displayed only in the drawing. Its purpose is to help you avoid accidentally overwriting parameters and formulas when the dimension name format is set to Value or Name, which suppresses the display of the parameters and formulas.

**Quick Reference**

**DIMCONSTRAINT**

Applies dimensional constraints to selected objects or points on objects, or converts associative dimensions to dimensional constraints.

**PARAMETERS (-PARAMETERS)**

Controls the associative parameters used in the drawing.

**Control Geometry with Parameters**

You can create and manage custom parameters that can be used with dimensional constraints (dynamic and annotational).

The -PARAMETERS command allows you to do the following operations:

- Create a new parameter
- Edit the expression of a parameter
- Rename a parameter
- Delete a parameter from the drawing
- List all the parameters in the current drawing
Use Operators in Expressions

Dimensional constraint parameters and user variables support the following operators within expressions:

<table>
<thead>
<tr>
<th>Operator</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>+</td>
<td>Addition</td>
</tr>
<tr>
<td>-</td>
<td>Subtraction or unary negation</td>
</tr>
<tr>
<td>%</td>
<td>Floating point modulo</td>
</tr>
<tr>
<td>*</td>
<td>Multiplication</td>
</tr>
<tr>
<td>/</td>
<td>Division</td>
</tr>
<tr>
<td>^</td>
<td>Exponentiation</td>
</tr>
<tr>
<td>(</td>
<td>Parenthesis, expression delimiter</td>
</tr>
<tr>
<td>.</td>
<td>Decimal separator</td>
</tr>
</tbody>
</table>

NOTE With imperial units, the a minus or dash (-) symbol is treated as a unit separator rather than a subtraction operation. To specify subtraction, include at least one space before or after the minus sign. For example, to subtract 9" from 5', enter 5'-9" rather than 5'-9".

Understand Precedence in Expressions

Expressions are evaluated according to the following standard mathematical rules of precedence:

1. Expressions in parentheses first, starting with the innermost set
2. Operators in standard order: (1) unary negation, (2) exponents, (3) multiplication and division, and (4) addition and subtraction
3. Operators of equal precedence from left to right
## Functions Supported in Expressions

The following functions are available for use in expressions:

<table>
<thead>
<tr>
<th>Function</th>
<th>Syntax</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cosine</td>
<td>( \cos(expression) )</td>
</tr>
<tr>
<td>Sine</td>
<td>( \sin(expression) )</td>
</tr>
<tr>
<td>Tangent</td>
<td>( \tan(expression) )</td>
</tr>
<tr>
<td>Arc cosine</td>
<td>( \arccos(expression) )</td>
</tr>
<tr>
<td>Arc sine</td>
<td>( \arcsin(expression) )</td>
</tr>
<tr>
<td>Arc tangent</td>
<td>( \arctan(expression) )</td>
</tr>
<tr>
<td>Hyperbolic cosine</td>
<td>( \cosh(expression) )</td>
</tr>
<tr>
<td>Hyperbolic sine</td>
<td>( \sinh(expression) )</td>
</tr>
<tr>
<td>Hyperbolic tangent</td>
<td>( \tanh(expression) )</td>
</tr>
<tr>
<td>Arc hyperbolic cosine</td>
<td>( \text{acosh}(expression) )</td>
</tr>
<tr>
<td>Arc hyperbolic sine</td>
<td>( \text{asinh}(expression) )</td>
</tr>
<tr>
<td>Arc hyperbolic tangent</td>
<td>( \text{atanh}(expression) )</td>
</tr>
<tr>
<td>Square root</td>
<td>( \sqrt(expression) )</td>
</tr>
<tr>
<td>Signum function (-1,0,1)</td>
<td>( \text{sign}(expression) )</td>
</tr>
<tr>
<td>Round to nearest integer</td>
<td>( \text{round}(expression) )</td>
</tr>
<tr>
<td>Truncate decimal</td>
<td>( \text{trunc}(expression) )</td>
</tr>
<tr>
<td>Round down</td>
<td>( \text{floor}(expression) )</td>
</tr>
<tr>
<td>Round up</td>
<td>( \text{ceil}(expression) )</td>
</tr>
<tr>
<td>Function</td>
<td>Syntax</td>
</tr>
<tr>
<td>---------------------------</td>
<td>-------------------------</td>
</tr>
<tr>
<td>Absolute value</td>
<td>\texttt{abs(expression)}</td>
</tr>
<tr>
<td>Largest element in array</td>
<td>\texttt{max(expression1;expression2)}</td>
</tr>
<tr>
<td>Smallest element in array</td>
<td>\texttt{min(expression1;expression2)}</td>
</tr>
<tr>
<td>Degrees to radians</td>
<td>\texttt{d2r(expression)}</td>
</tr>
<tr>
<td>Radians to degrees</td>
<td>\texttt{r2d(expression)}</td>
</tr>
<tr>
<td>Logarithm, base e</td>
<td>\texttt{ln(expression)}</td>
</tr>
<tr>
<td>Logarithm, base 10</td>
<td>\texttt{log(expression)}</td>
</tr>
<tr>
<td>Exponent, base e</td>
<td>\texttt{exp(expression)}</td>
</tr>
<tr>
<td>Exponent, base 10</td>
<td>\texttt{exp10(expression)}</td>
</tr>
<tr>
<td>Power function</td>
<td>\texttt{pow(expression1;expression2)}</td>
</tr>
<tr>
<td>Random decimal, 0-1</td>
<td>\texttt{Random}</td>
</tr>
</tbody>
</table>

In addition to these functions, the constants Pi and e are also available for use in expressions.

**Quick Reference**

**DIMCONSTRAINT**

Applies dimensional constraints to selected objects or points on objects, or converts associative dimensions to dimensional constraints.

**PARAMETERS (-PARAMETERS)**

Controls the associative parameters used in the drawing.
Define and Reference Blocks
A block is one or more objects combined to create a single object. Blocks help you reuse objects in the same drawing or in other drawings.

**Overview of Blocks**

**How Blocks Are Stored and Referenced**

Every drawing file has a block definition table that stores all block definitions, which consist of all information associated with the block. It is these block definitions that are referenced when you insert blocks in your drawing.

Each rectangle below represents a separate drawing file and is divided into two parts:

- The block definition table
- The objects in the drawing
When you insert a block you are inserting a block reference. The information is not copied from the block definition to the drawing area. Instead, a link is established between the block reference and the block definition. Therefore, if the block definition is changed, all references are updated automatically.

Use PURGE to remove unused block definitions from a drawing.

**Blocks and Layers**

A block can be composed of objects drawn on several layers with various colors, linetypes, and lineweight properties. Although a block is always inserted on the current layer, the block reference preserves information about the original layer, color, and linetype properties of the objects that are contained in the block. You can control whether objects in a block retain their original properties or inherit their properties from the current layer, color, linetype, or linewidth settings.

**Annotative Blocks**

You can also create blocks. For more information about creating and working with an annotative blocks, see Create Annotative Blocks and Attributes on page 667.

**See also:**

- Scale Annotations on page 656
- Create Annotative Blocks and Attributes on page 667

**Quick Reference**

B
c

Keats a b l o c k d i f f e n t i a l f r o m s e l e c t e d o b j e c t s.

P

U R G E

R e m o v e s u n u s e d i t e m s, s u c h a s b l o c k d e f i n i t i o n s a n d l a y e r s, f r o m t h e d r a w i n g.

W

B L O C K

W r i t e s o b j e c t s o r a b l o c k t o a n e w d r a w i n g f i l e.

M

A X S O R T

Set s t h e m a x i m u m n u m b e r o f s y m b o l n a m e s o r b l o c k n a m e s s o r t e d b y l i s t i n g c o m m a n d s.
Insert Blocks

When you insert a block, you create a block reference and specify its location, scale, and rotation.

Scale Block References

You can specify the scale of a block reference using different $X$, $Y$, and $Z$ values.

A block that uses different drawing units than the units specified for the drawing is automatically scaled by a factor equivalent to the ratio between the two units.

Edit Attribute Values

If you insert a block reference that includes editable attributes, you can change the values of these attributes in the Edit Attributes dialog box or at the Command prompt while inserting the block or using one of the following ways after a block is inserted:

- Double-click a block and use the Enhanced Attribute Editor (EATTEDIT command).
- Use the Edit Attribute dialog box (ATTEDIT command).
- Enter-ATTEDIT at the Command prompt.
- Select a block and edit attribute values under Attributes in the Properties Inspector.

Insert a Drawing File as a Block

When you insert an entire drawing file into another drawing, the drawing information is copied into the block table of the current drawing as a block definition. Subsequent insertions reference the block definition with different position, scale, and rotation settings, as shown in the following illustration.
Xrefs contained in a drawing you insert may not be displayed properly unless the xref was previously inserted or attached to the destination drawing.

**Insert Blocks from Block Libraries**

You can insert one or more block definitions from an existing drawing file into your current drawing file. Choose this method when retrieving blocks from block library drawings. A block library drawing contains block definitions of symbols with similar functions. These block definitions are stored together in a single drawing file for easy accessibility and management.

**Insert Blocks with Content Palette**

Use the Content palette to insert blocks from the current drawing or from a library. Drag and drop, or double-click a block to insert it into the current drawing.

**See also:**

- [Create Drawing Files for Use as Blocks](#) on page 414
- [Overview of Blocks](#) on page 405
- [Add Text and Blocks to Tables](#) on page 762
**Quick Reference**

**CONTENT**
Opens the Content palette.

**DIVIDE**
Creates evenly spaced point objects or blocks along the length or perimeter of an object.

**INSERT**
Inserts a block or drawing into the current drawing.

**MEASURE**
Creates point objects or blocks at measured intervals along the length or perimeter of an object.

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

**INSNAME**
Sets a default block name for the INSERT command.

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

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

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

**Work with Dynamic Blocks in Drawings**
A dynamic block reference can be changed in a drawing while you work.

**Overview of Dynamic Blocks**
Dynamic block references contain grips or custom properties that change the way the reference is displayed in the drawing after it is inserted. For example,
a dynamic block reference of a door can change size after you insert the block reference into your drawing. Dynamic blocks allow you to insert one block that can change shape, size, or configuration, instead of inserting one of many static block definitions.

NOTE Dynamic blocks are not supported in AutoCAD 2011 for Mac. When a drawing containing dynamic blocks is opened, the blocks will function like standard blocks.

Quick Reference

ATTSYNC
Updates block references with new and changed attributes from a specified block definition.

INSERT
Inserts a block or drawing into the current drawing.

PROPERTIES
Controls properties of existing objects.

RESETBLOCK
Resets one or more dynamic block references to the default values of the block definition.
BTMARKDISPLAY
Controls whether or not value set markers are displayed for dynamic block references.

GRIPTIPS
Controls the display of grip tips and Ctrl-cycling tooltips.

Remove Block Definitions
To reduce the size of a drawing, you can remove unused block definitions. You can remove a block reference from your drawing by erasing it; however, the block definition remains in the drawing's block definition table.

To remove unused block definitions and decrease the drawing size, use PURGE at any time in your drawing session.

All references to a block must be erased before you can purge the block definition.

See also:
■ Overview of Blocks on page 405

Quick Reference
PURGE
Removes unused items, such as block definitions and layers, from the drawing.
Create and Modify Blocks

A block definition is a set of objects that are grouped together as one named object with a base point and unique properties.

Define Blocks

You create blocks by associating objects and giving them a name.

Create Blocks Within a Drawing

After you define a block in a drawing, you can insert a block reference in the drawing as many times as necessary. Use this method to create blocks quickly.

Each block definition includes a block name, one or more objects, the coordinate values of the base point to be used for inserting the block, and any associated attribute data.

The base point is used as a reference for positioning the block when you insert it. Suppose you specify that the base point is at the lower-left corner of an object in the block. Later, when you insert the block, you are prompted for an insertion point. The block base point is aligned at the insertion point you specified.

The block definition in the illustration comprises a name, PLUG_VALVE, four lines, and a base point at the intersection of the two diagonal lines. For an explanation of the schematic representation shown, see Overview of Blocks on page 405.
The illustration shows a typical sequence for creating a block definition within a drawing.

You can also use the Block Editor to create blocks that are saved within a drawing.

See also:

- Overview of Blocks on page 405

Quick Reference

BLOCK

Creates a block definition from selected objects.

Create Drawing Files for Use as Blocks

You can create drawing files for the purpose of inserting them into other drawings as blocks. Individual drawing files are easy to create and manage as the source of block definitions. Collections of symbols can be stored as individual drawing files and grouped in folders.
Create a New Drawing File

You have two methods for creating drawing files:

- Create and save a complete drawing file using SAVE or SAVEAS.
- Create and save only selected objects from your current drawing to a new
  drawing using EXPORT or WBLOCK.

With either method, you create an ordinary drawing file that can be inserted
as a block into any other drawing file. Using WBLOCK is recommended when
you need to create several versions of a symbol as separate drawing files, or
when you want to create a drawing file without leaving the current drawing.

Change the Base Point of Drawings to Be Used as Blocks

By default, the WCS (world coordinate system) origin (0,0,0) is used as the
base point for drawing files inserted as blocks. You can change the base point
by opening the original drawing and using BASE to specify a different base
point for insertion. The next time you insert the block, the new base point is
used.

Update Changes in the Original Drawing

If you change the original drawing after inserting it, the changes have no
effect on the current drawing. If you expect the original drawing to change,
and you want the changes to be reflected in the current drawing, you may
want to attach it as an external reference instead of inserting it as a block. For
more information about external references, see Reference Other Drawing
Files on page 881.

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.
Quick Reference

CONTENT
Opens the Content palette.

BASE
Sets the insertion base point for the current drawing.

BLOCK
Creates a block definition from selected objects.

EXPORT
Saves the objects in a drawing to a different file format.

INSERT
Inserts a block or drawing into the current drawing.

OOPS
Restores erased objects.

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

Control the Color and Linetype Properties in Blocks

Assign Color and Linetype Properties
Generally when you insert a block, the color, linetype, and lineweight of objects in the block retain their original settings regardless of the current settings in the drawing. However, you can create blocks with objects that inherit the current color, linetype, and lineweight settings. These objects have floating properties.

You have three choices for how the color, linetype, and lineweight properties of objects are treated when a block reference is inserted.

- Objects in the block do not inherit color, linetype, and lineweight properties from the current settings. The properties of objects in the block do not change regardless of the current settings.
  For this choice, it is recommended that you set the color, linetype, and lineweight properties individually for each object in the block definition:
do not use BYBLOCK or BYLAYER color, linetype, and lineweight settings when creating these objects.

- Objects in the block inherit color, linetype, and lineweight properties from the color, linetype, and lineweight assigned to the current layer only. For this choice, before you create objects to be included in the block definition, set the current layer to 0, and set the current color, linetype, and lineweight to BYLAYER.

- Objects inherit color, linetype, and lineweight properties from the current color, linetype, and lineweight that you have set explicitly, that is, that you have set to override the color, linetype, or lineweight assigned to the current layer. If you have not explicitly set them, then these properties are inherited from the color, linetype, and lineweight assigned to the current layer.

  For this choice, before you create objects to be included in the block definition, set the current color or linetype to BYBLOCK.

<table>
<thead>
<tr>
<th>If you want objects in a block to</th>
<th>Create objects on these layers</th>
<th>Create objects with these properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>Retain original properties</td>
<td>Any but 0 (zero)</td>
<td>Any but BYBLOCK or BYLAYER</td>
</tr>
<tr>
<td>Inherit properties from the current layer</td>
<td>0 (zero)</td>
<td>BYLAYER</td>
</tr>
<tr>
<td>Inherit individual properties first, then layer properties</td>
<td>Any</td>
<td>BYBLOCK</td>
</tr>
</tbody>
</table>

Floating properties also apply to nested blocks when the nested block references and the objects they contain use the settings required for floating properties.

**Change the Color and Linetype in a Block**

You can change the color and linetype of the objects within a block only if the objects in that block were created with floating properties.

If a block was not created using objects with floating color and linetype properties, the only way to change these properties is to redefine the block.

**See also:**

- **Control the Properties of Objects** on page 161
Quick Reference

COLOR
Sets the color for new objects.

LAYER
Manages layers and layer properties.

LINETYPE
Loads, sets, and modifies linetypes.

PROPERTIES
Controls properties of existing objects.

Nest Blocks

The only restriction on nested blocks is that you cannot insert blocks that reference themselves.

You can apply geometric constraints and constraint parameters to nested objects in blocks. AutoCAD for Mac detects the nested entity or valid constraint point for the nested entity regardless of the nesting level of the object.

NOTE: Constraints can only be applied between nested objects in the block and objects in the drawing file, not between pairs of nested objects in the block reference.
When a block definition is redefined, AutoCAD for Mac will re-evaluate the constraints between geometry in the drawing and the nested geometry in the block references. The drawing will then be updated appropriately. If a constraint cannot be resolved as a result of the change to the block definition, then the constraint is removed and an unresolved constraints message is displayed at the command line.

See also:
- Overview of Blocks on page 405

Quick Reference

BLOCK

Creates a block definition from selected objects.

Create Block Libraries

A block library is a collection of block definitions stored in a single drawing file. You can use block libraries supplied by Autodesk or other vendors or create your own.

You can organize a set of related block definitions by creating the blocks in the same drawing file. Drawing files used this way are called block, or symbol, libraries. These block definitions can be inserted individually into any drawing that you are working on. Block library drawings are not different from other drawing files except in how they are used.

When you use BLOCK to define each block definition in the block library drawing, you can include a short description of the block.

Optionally, you can also document each block definition by inserting it in the drawing area of the library drawing. In addition to the block geometry, you can include text that provides the block name, the date of creation, the date of the last modification, and any special instructions or conventions. This creates a visual index of the blocks in the block library drawing.
Use the Content palette to view and insert block definitions from the current or an existing drawing. Insert a block from the Content palette does not overwrite an existing block definition in a drawing with one that comes from another drawing.

**Quick Reference**

**BLOCK**

Creates a block definition from selected objects.

**Attach Data to Blocks (Block Attributes)**

You can attach information to blocks and later extract the information to create a bill of materials or other report.

**Overview of Block Attributes**

An attribute is a label or tag that attaches data to a block. Examples of data that might be contained in an attribute are part numbers, prices, comments, and owners’ names. The tag is equivalent to a column name in a database table. The following illustration shows a block with four attributes: type, manufacturer, model, and cost.
The attributes in the illustration are single-line attributes. You can also create multiple-line attributes to store data such as addresses and descriptions.

Attribute information extracted from a drawing can be used in a spreadsheet or database to produce a parts list or a bill of materials. You can associate more than one attribute with a block, provided that each attribute has a different tag.

Attributes also can be "invisible." An invisible attribute is not displayed or plotted; however, the attribute information is stored in the drawing file and can be written to an extraction file for use in a database program.

Whenever you insert a block that has a variable attribute, you are prompted to enter data to be stored with the block. Blocks can also use constant attributes, attributes whose values do not change. Constant attributes do not prompt you for a value when you insert the block.

You can also create attributes. For more information about creating and working with an annotative attributes, see Create Annotative Blocks and Attributes on page 667.

See also:

- Modify a Block Attribute Definition on page 434
- Modify the Data in Block Attributes on page 433
- Scale Annotations on page 656
Quick Reference

ATTDEF
   Creates an attribute definition for storing data in a block.

ATTDISP
   Controls the visibility overrides for all block attributes in a drawing.

ATTEDIT
   Changes attribute information in a block.

ATTIPEDIT
   Changes the textual content of an attribute within a block.

DDEDIT
   Edits single-line text, dimension text, attribute definitions, and feature control frames.

PROPERTIES
   Controls properties of existing objects.

AFLAGS
   Sets options for attributes.

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

ATTIPE
   Controls the display of the in-place editor used to create multiline attributes.

ATTMODE
   Controls display of attributes.

ATTMULTI
   Controls whether multiline attributes can be created.

ATTREQ
   Controls whether INSERT uses default attribute settings during insertion of blocks.
Define Block Attributes

The characteristics include the tag, which is a name that identifies the attribute, the prompt displayed when you insert the block, value information, text formatting, location within the block, and any optional modes (Invisible, Constant, Verify, Preset, Lock Position, and Multiple Lines).

If you plan to extract the attribute information for use in a parts list, you may want to keep a list of the attribute tags you have created. You will need this tag information later when you create the attribute template file.

Choose Attribute Modes

Attribute modes control the behavior of attributes in blocks. For example, you can control

- Whether an attribute is visible or invisible in the drawing
- Whether an attribute has a constant value, such as a part number
- Whether the attribute can be moved relative to the rest of the block
- Whether the attribute is a single-line attribute or a multiple-line attribute

If an attribute has a constant value, you will not be prompted for its value when you insert the block. If an attribute has a variable value, such as the asset number of a computer, you will be prompted when you insert the block.

Understand Single-Line and Multiple-Line Attributes

There are several differences between single-line and multiple-line attributes.

- Single-line attributes are limited to 255 characters from the user interface.
- Multiple-line attributes provide more formatting options than single-line attributes.
- When editing single-line and multiple line attributes, different editors are displayed.
- Multiple line attributes display four grips similar to MTEXT objects, while single-line attributes display only one grip.
- When a drawing is saved to AutoCAD 2007 or earlier, a multiple-line attribute is converted to several single-line attributes, one for every line of text in the original multiple-line attribute. If the drawing file is opened in
the current release, these single line attributes are automatically merged back into a multiple-line attribute.

**NOTE** If a multiple-line attribute makes a round trip to an earlier release, the differences between these two types of attributes might result in truncating very long lines of text and loss of formatting. However, before any characters are truncated, AutoCAD for Mac displays a message box that lets you cancel the operation.

### Correct Mistakes in Block Attribute Definitions

If you make a mistake, you can use the Properties Inspector palette or DDDEDIT to make limited changes to an attribute definition before it is associated with a block. If you need to make more extensive changes, delete the attribute definition and create a new one.

### Attach Attributes to Blocks

After you create one or more attribute definitions, you attach the attributes to a block when you define or redefine that block. When you are prompted to select the objects to include in the block definition, include in the selection set any attributes you want to attach to the block.

To use several attributes together, define them and then include them in the same block. For example, you can define attributes tagged "Type," "Manufacturer," "Model," and "Cost," and then include them in a block called CHAIR.

![Attributes Example](image)

Usually, the order of the attribute prompts is the same as the order in which you selected the attributes when you created the block. However, if you used crossing or window selection to select the attributes, the order of the prompts is the reverse of the order in which you created attributes. You can use the Block Attribute Manager to change the order in which you are prompted for attribute information when you insert the block reference.

When you open a block definition in the Block Editor, you can use the Attribute Order dialog box (BATTORDER command) to change the order in
which you are prompted for attribute information when you insert the block reference.

**Use Attributes Without Attaching Them to Blocks**

Stand-alone attributes can also be created. Once attributes have been defined, and the drawing is saved, this drawing file can be inserted into another drawing. When the drawing is inserted, you are prompted for the attribute values.

**Quick Reference**

**ATTDEF**
Creates an attribute definition for storing data in a block.

**ATTDISP**
Controls the visibility overrides for all block attributes in a drawing.

**BATTORDER**
Specifies the order of attributes for a block.

**DDEDIT**
Edits single-line text, dimension text, attribute definitions, and feature control frames.

**PROPERTIES**
Controls properties of existing objects.

**AFLAGS**
Sets options for attributes.

**ATTIPE**
Controls the display of the in-place editor used to create multiline attributes.

**ATTMULTI**
Controls whether multiline attributes can be created.

**Extract Data from Block Attributes**

Extracting attribute information is an easy way to produce a schedule or bill of materials directly from your drawing data. For example, a facilities drawing...
might contain blocks representing office equipment. If each block has attributes identifying the model and manufacturer of the equipment, you can generate a report that estimates the cost of the equipment.

**Output to a File**

If you save the data to an external file, the comma-separated (CDF), space delimited (SDF), and data extraction (DXX) formats are available. Your can use the following commands to extract attribute information:

- **ATTEXT** - Extracts attribute information using a template file that describes which attribute values to extract.
- **EXPORT** - Extracts attribute information from a drawing into a DXX file format. DXX is a file format similar to DXF except it only contains attribute information.

**Quick Reference**

**ATTEXT**

Extracts attribute data, informational text associated with a block, into a file.

**EXPORT**

Saves the objects in a drawing to a different file format.

**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 for Mac, 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.

```plaintext
BL:NAME Cwww000 (Block name)
BL:LEVEL Nwww000 (Block nesting level)
BL:X Nwwwddd(X coordinate of block insertion point)
BL:Y Nwwwddd(Y coordinate of block insertion point)
BL:Z Nwwwddd(Z coordinate of block insertion point)
BL:NUMBER Nwww000 (Block counter; the same for MINSERT)
BL:HANDLE Cwww000 (Block handle; the same for MINSERT)
BL:LAYER Cwww000 (Block insertion layer name)
BL:ORIENT Nwwwddd(Block rotation angle)
BL:XSCALE Nwwwddd(X scale factor)
BL:YSCALE Nwwwddd(Y scale factor)
BL:ZSCALE Nwwwddd(Z scale factor)
BL:XEXTRUDE Nwwwddd(X component of block extrusion direction)
BL:YEXTRUDE Nwwwddd(Y component of block extrusion direction)
BL:ZEXTRUDE Nwwwddd(Z component of block extrusion direction)
numericNwwwddd (Numeric attribute tag)
characterCwww000 (Character attribute tag)
```

The template file can include any or all of the BL:xxxxx field names listed, but must include at least one attribute tag field. The attribute tag fields
determine which attributes, hence which blocks, are included in the attribute extraction file. If a block contains some, but not all, of the specified attributes, the values for the absent ones are filled with blanks or zeros, depending on whether the field is a character field or a numeric field.

Comments should not be included in an attribute template file.

The illustration and table show an example of the type of information you're likely to extract, including block name, manufacturer, model number, and cost.

<table>
<thead>
<tr>
<th>Field</th>
<th>Character or Numeric data</th>
<th>Maximum Field Length</th>
<th>Decimal Places</th>
</tr>
</thead>
<tbody>
<tr>
<td>Block name</td>
<td>C</td>
<td>040</td>
<td>000</td>
</tr>
<tr>
<td>Manufacturer</td>
<td>C</td>
<td>006</td>
<td>000</td>
</tr>
<tr>
<td>Model</td>
<td>C</td>
<td>015</td>
<td>000</td>
</tr>
<tr>
<td>Cost</td>
<td>N</td>
<td>006</td>
<td>002</td>
</tr>
</tbody>
</table>

You can create any number of template files, depending on how you'll use the data. Each line of a template file specifies one field to be written in the attribute extraction file.

Follow these additional guidelines:

- Be sure to place a space between the attribute tag and the character or numeric data. Use Spacebar, not Tab, to enter the space.
- Press Enter at the end of each line, including the last line.
- Each attribute extraction template file must include at least one attribute tag field, but the same field can appear only once in the file.

The following is a sample template file.

BL:NAME C008000 (Block name, 8 characters)
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:

'DESC', 120.0, 49.5, 'ACME INDUST.', '51-793W', 379.95
'CHAIR', 122.0, 47.0, 'ACME INDUST.', '34-902A', 199.95
'DESC', -77.2, 40.0, 'TOP DRAWER INC.', 'X-52-44', 249.95

By default, character fields are enclosed with single quotes (apostrophes). The default field delimiter is a comma. The following two template records can be used to override these defaults:

C:QUOTE c (Character string delimiter)
C:DELIM c (Field delimiter)

The first nonblank character following the C:QUOTE or C:DELIM field name becomes the respective delimiter character. For example, if you want to enclose character strings with double quotes, include the following line in your attribute extraction template file:

C:QUOTE "

The quote delimiter must not be set to a character that can appear in a character field. Similarly, the field delimiter must not be set to a character that can appear in a numeric field.

If you specified an SDF format using the sample template, the file might be similar to the following example.

<table>
<thead>
<tr>
<th>(NAME)</th>
<th>(X)</th>
<th>(Y)</th>
<th>(SUPPLIER)</th>
<th>(MODEL)</th>
<th>(PRICE)</th>
</tr>
</thead>
<tbody>
<tr>
<td>DESK</td>
<td>120.0</td>
<td>49.5</td>
<td>ACME INDUST.</td>
<td>51-793W</td>
<td>379.95</td>
</tr>
<tr>
<td>CHAIR</td>
<td>122.0</td>
<td>47.0</td>
<td>ACME INDUST.</td>
<td>34-902A</td>
<td>199.95</td>
</tr>
<tr>
<td>DESK</td>
<td>-77.2</td>
<td>40.0</td>
<td>TOP DRAWER INC.</td>
<td>X-52-44</td>
<td>249.95</td>
</tr>
</tbody>
</table>

The order of the fields corresponds to the order of the fields in the template files. You can use these files in other applications, such as spreadsheets, and you can sort and manipulate the data as needed. See the documentation for your spreadsheet program for information about how to use data from other applications. If you open the file in a text editor or a word processor, you can paste the information back into the drawing as text.
Nested Blocks

The line BL:LEVEL in a template file reports the nesting level of a block reference. A block that is inserted in a drawing has a nesting level of 1. A block reference that is part of (nested within) another block has a nesting level of 2, and so on.

For a nested block reference, the \(X, Y, Z\) coordinate values, scale factors, extrusion direction, and rotation angle reflect the actual location, size, orientation, and rotation of the nested block in the world coordinate system.

In some complex cases, nested block references cannot be correctly represented with only two scale factors and a rotation angle, for example, if a nested block reference is rotated in 3D. When this happens, the scale factors and rotation angle in the extracted file record are set to zero.

Error Handling

If a field is not wide enough for the data that is to be placed in it, the data is truncated and the following message is displayed:

** Field overflow in record <record number>

This could happen, for example, if you have a BL:NAME field with a width of 8 characters and a block in your drawing has a name 10 characters long.

Quick Reference

ATTEXT

Extracts attribute data, informational text associated with a block, into a file.

Modify Blocks

You can modify a block definition or a block reference already inserted in the drawing.

Modify a Block Definition

You can redefine block definitions in your current drawing. Redefining a block definition affects both previous and future insertions of the block in the current drawing and any associated attributes.
There are two methods for redefining a block definition:

- Modify the block definition in the current drawing.
- Modify the block definition in the source drawing and reinsert it into the current drawing.

The method you choose depends on whether you want to make changes in the current drawing only or in a source drawing also.

**Modify a Block Definition in the Current Drawing**

To modify a block definition, follow the procedure to create a new block definition, but enter the name of the existing block definition. This replaces the existing block definition, and all the references to that block in the drawing are immediately updated to reflect the new definition.

To save time, you can insert and explode an instance of the original block and then use the resulting objects in creating the new block definition.

**Update a Block Definition That Originated from a Drawing File**

Block definitions created in your current drawing by inserting a drawing file are not updated automatically when the original drawing is modified. You can use INSERT to update a block definition from the drawing file.

**Update a Block Definition That Originated from a Library Drawing (Advanced)**

Content palette does not overwrite an existing block definition in a drawing with one that comes from another drawing. To update a block definition that came from a library drawing, use WBLOCK to create a separate drawing file from the library drawing block. Then, use INSERT to overwrite the block definition in the drawing that uses the block.

**NOTE** Block descriptions are stripped off when using INSERT. Use the Clipboard to copy and paste a block description displayed in the Define Block dialog box from one block definition to another.

**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 420

Quick Reference

BLOCK

Creates a block definition from selected objects.

EXPLODE

Breaks a compound object into its component objects.

PROPERTIES

Controls properties of existing objects.

Modify the Data in Block Attributes

You can use any of the following methods to edit the values of attributes attached to a block:

- Double-click the block to display the Enhanced Attributes Editor
- Press Ctrl and double-click the attribute to display the in-place editor
- Open the Properties Inspector palette and select the block

You can also change the location of attributes in a block using grips. With multiple-line attributes, you can also move grips to resize the width of the text.
NOTE If you press Ctrl and double-click an attribute that includes a hyperlink, the hyperlink opens the web page. To edit the attribute, use one of the other methods listed.

See also:
- Modify a Block Definition on page 431

**Quick Reference**

ATTEDIT
Changes attribute information in a block.

ATTIPEDIT
Changes the textual content of an attribute within a block.

ATTSYNC
Updates block references with new and changed attributes from a specified block definition.

BATTMAN
Manages the attributes for a selected block definition.

EATTEDIT
Edits attributes in a block reference.

ATTIPE
Controls the display of the in-place editor used to create multiline attributes.

ATTMULTI
Controls whether multiline attributes can be created.

**Modify a Block Attribute Definition**

You can modify attributes in block definitions with the Block Attribute Manager. For example, you can modify the following:

- Properties that define how values are assigned to an attribute and whether or not the assigned value is visible in the drawing area
- Properties that define how attribute text is displayed in the drawing area
Properties that define the layer that the attribute is on and the attribute line's color, weight, and type

By default, attribute changes you make are applied to all existing block references in the current drawing.

Changing the attribute properties of existing block references does not affect the values assigned to those blocks. For example, in a block containing an attribute whose tag is Cost and value is 19.99, the 19.99 value is unaffected if you change the tag from Cost to Unit Cost.

Updating attributes with duplicate tag names can lead to unpredictable results. Use the Block Attribute Manager to find duplicate tags and change tag names.

If constant attributes or nested attributed blocks are affected by your changes, use REGEN to update the display of those blocks in the drawing area.

Change the Prompt Order for Attribute Values

When you define a block, the order in which you select the attributes determines the order in which you are prompted for attribute information when you insert the block. You can use the Block Attribute Manager to change the order of prompts that request attribute values.

Remove Block Attributes

You can remove attributes from block definitions and from all existing block references in the current drawing. Attributes removed from existing block references do not disappear in the drawing area until you regenerate the drawing using REGEN.

You cannot remove all attributes from a block; at least one attribute must remain. If you need to remove all attributes, redefine the block.

Update Block References

You can update attributes in all block references in the current drawing with changes you made to the block definition. For example, you may have used the Block Attribute Manager to modify attribute properties in several block definitions in your drawing but elected not to automatically update existing block references when you made the changes. Now that you are satisfied with the attribute changes you made, you can apply those changes to all blocks in the current drawing.

You can also use ATTSYNC to update attribute properties in block references to match their block definition, or to update a block instance after you redefine a block attribute using BLOCK, -BLOCK, or BEDIT.
Updating attribute properties in block references does not affect any values that have been assigned to those attributes.

**Edit Attributes in a Block Reference**

You can select an attribute in a block reference and use the Properties Inspector palette to change its properties, or you can use the Enhanced Attribute Editor to modify all the attributes in a selected block reference.

See also:
- Define Block Attributes on page 423
- Modify a Block Definition on page 431

**Quick Reference**

**ATTIPEDIT**
Changes the textual content of an attribute within a block.

**ATTSYNC**
Updates block references with new and changed attributes from a specified block definition.

**BATTMAN**
Manages the attributes for a selected block definition.

**EATTEDIT**
Edits attributes in a block reference.

**ATTIPE**
Controls the display of the in-place editor used to create multiline attributes.

**ATTMULTI**
Controls whether multiline attributes can be created.
Disassemble a Block Reference (Explode)

If you need to modify one or more objects within a block separately, you can disassemble, or explode, the block reference into its component objects. After making the changes, you can

- Create a new block definition
- Redefine the existing block definition
- Leave the component objects uncombined for other uses

When you explode a block reference, the block reference is disassembled into its component objects; however, the block definition still exists in the drawing for insertion later.

You can automatically explode block references as you insert them by selecting the Explode option in the Insert Block dialog box.

Quick Reference

EXPLODE

Breaks a compound object into its component objects.

XPLODE

Breaks a compound object into its component objects.
Work with 3D Models
Create 3D Models

Use 3D models to help you visualize and test your designs.

Overview of 3D Modeling

AutoCAD 3D modeling allows you to create drawings using solid, surface, and mesh objects.

Solid, surface, and mesh objects offer different functionality, that, when used together, offer a powerful suite of 3D modeling tools. For example, you can convert a primitive solid to a mesh to take advantage of mesh creasing and smoothing. You can then convert the model to a surface to take advantage of associativity and NURBS modeling.

Solid Modeling

A solid model is an enclosed 3D body that has properties such as mass, volume, center of gravity, and moments of inertia.

Start with primitive solids such as cones, boxes, cylinders, and pyramids and modify and recombine them to create new shapes. Or draw a custom polysolid.
extrusion and use various sweeping operations to create solids from 2D curves and lines.

**Surface Modeling**

A surface model is a thin shell that does not have mass or volume. AutoCAD offers two types of surfaces: procedural and NURBS. Use procedural surfaces to take advantage of associative modeling, and use NURBS surfaces to take advantage of sculpting with control vertices.

A typical modeling workflow is to create a basic model using mesh, solids, and procedural surfaces, and then convert them to NURBS surfaces. This allows you to utilize not only the unique tools and primitive shapes offered by solids and meshes, but also the shaping capabilities provided by surfaces - associative modeling and NURBS modeling.

You create surface models using some of the same tools that you use for solid models: sweeping, lofting, extruding, and revolving. You can also create surfaces by blending, patching, offsetting, filleting, and extending other surfaces.
Mesh Modeling

A mesh model consists of vertices, edges, and faces that use polygonal representation (including triangles and quads) to define a 3D shape.

Unlike solid models, mesh has no mass properties. However, as with 3D solids, you can create primitive mesh forms such as boxes, cones, and pyramids, starting in AutoCAD 2010. You can modify mesh models in ways that are not available for 3D solids or surfaces. For example you can apply creases, splits, and increasing levels of smoothness. You can drag mesh subobjects (faces, edges, and vertices) to deform the object. To achieve more granular results, you can refine the mesh in specific areas before modifying it.

Use mesh models to provide the hiding, shading, and rendering capabilities of a solid model without the physical properties such as mass, moments of inertia, and so on.

Advantages of 3D Modeling

Modeling in 3D has several advantages. You can

- View the model from any vantage point
- Generate reliable standard and auxiliary 2D views automatically
- Create sections and 2D drawings
- Remove hidden lines and do realistic shading
- Check interferences and perform engineering analysis
- Add lighting and create realistic rendering
- Navigate through the model
- Use the model to create an animation
- Extract manufacturing data

See also:
- Enter 3D Coordinates on page 205
- Specify Workplanes in 3D (UCS) on page 214
- Use the Dynamic UCS with Solid Models on page 217

Quick Reference

Commands

BOX
  Creates a 3D solid box.
CONE
  Creates a 3D solid cone.
CONVTOMESH
  Converts 3D objects such as polygon meshes, surfaces, and solids to mesh objects.
CONVTONURBS
  Converts 3D solids and surfaces into NURBS surfaces.
CONVTOSOLID
  Converts 3D meshes and polylines and circles with thickness to 3D solids.
CONVTOSURFACE
  Converts objects to 3D surfaces.
CYLINDER
  Creates a 3D solid cylinder.
EXTRUDE
  Creates a 3D solid or surface by extending the dimensions of an object.
LOFT
Creates a 3D solid or surface in the space between several cross sections.

MESH
Creates a 3D mesh primitive object such as a box, cone, cylinder, pyramid, sphere, wedge, or torus.

POLYSOLID
Creates a 3D wall-like polysolid.

PLANESURF
Creates a planar surface.

PYRAMID
Creates a 3D solid pyramid.

REVOLVE
Creates a 3D solid or surface by sweeping an object around an axis.

SPHERE
Creates a 3D solid sphere.

SURFBLEND
Creates a continuous blend surface between two existing surfaces.

SURFNTERWORK
Creates a surface in the space between several curves in the U and V directions (including surface and solid edge subobjects).

SURFOFFSET
Creates a parallel surface a specified distance from the original surface.

SURFPATCH
Creates a new surface by fitting a cap over a surface edge that forms a closed loop.

SWEEP
Creates a 3D solid or surface by sweeping a 2D or 3D object or subobject along a path.

TORUS
Creates a donut-shaped 3D solid.
UCS
Manages user coordinate systems.

WEDGE
Creates a 3D solid wedge.

System Variables
DELOBJ
Controls whether geometry used to create 3D objects is retained or deleted.

PSOLWIDTH
Controls the default width for a swept solid object created with the POLYSOLID command.

PSOLHEIGHT
Controls the default height for a swept solid object created with the POLYSOLID command.

SUBOBJSELECTIONMODE
Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

SURFACEASSOCIATIVITY
Controls whether surfaces maintain a relationship with the objects from which they were created.

SURFACEMODELINGMODE
Controls whether surfaces are created as procedural surfaces or NURBS surfaces.

Create Solids and Surfaces from Lines and Curves
Use lines and curves to extrude, sweep, loft, and revolve 3D solids, surfaces, and NURBS surfaces.

Overview of Creating Solids and Surfaces
Understand the differences between creating solids and surfaces with the EXTRUDE, SWEEP, LOFT, and REVOLVE commands.
See also:

- Create a Solid or Surface by Extruding on page 450
- Create a Solid or Surface by Sweeping on page 453
- Create a Solid or Surface by Lofting on page 456
- Create a Solid or Surface by Revolving on page 459

Create Surfaces or Solids With the Same Commands

When you extrude, sweep, loft, and revolve curves, you can create both solids and surfaces. Open curves always create surfaces, but closed curves can create either solids or surfaces depending on the situation.

If you select a closed curve and click EXTRUDE, SWEEP, LOFT, and REVOLVE on the ribbon, you create:

- A solid if the Mode option is set to Solid.
- A surface if the Mode option is set to Surface.
  - A procedural surface if the SURFACEMODELINGMODE system variable is set to 0.
  - A NURBS surface if the SURFACEMODELINGMODE system variable is set to 1.
  - An associative surface if the SURFACEASSOCIATIVITY system variable is on.

In this illustration, the same profile creates a solid (left), a procedural surface (middle), and a NURBS surface (right).
Geometry That Can Be Used As Profiles and Guide Curves

The curves that you use as profile and guide curves when you extrude, sweep, loft, and revolve can be:

- Open or closed
- Planar or non-planar
- Solid and surface edge subobjects
- A single object (to extrude multiple lines, convert them to a single object with the JOIN command)
- A single region (to extrude multiple regions, convert them to a single object with the REGION command)

Example: Use Splines to Create 3D NURBS Surfaces

Splines are one of the many 2D object types that can be lofted, extruded, swept, and revolved to create NURBS surfaces. Other 2D objects that can be used include lines, polylines, arcs, and circles. Splines, however, are the only 2D object customized to create NURBS surfaces. Because they allow you to adjust tolerance, degree, and tangency, they are better suited than other types of 2D profiles (line, plines, circles) for surface modeling.
Many of the same commands used with NURBS surfaces, can also be used with CV splines. These include:

- CVADD
- CVREMOVE
- CVREBUILD
- CVSHOW
- CVHIDE

For more information, see Create Solids and Surfaces from Lines and Curves on page 446.

Create Associative Surfaces

Surfaces can be associative while solids cannot. If surface associativity is on when a surface is created, it maintains a relationship with the curve from which it is was generated (even if the curve is the subobject of another solid or surface). If the curve is reshaped, the surface profile automatically updates. See Create Associative Surfaces on page 505.

NOTE To modify a surface that is associative, you must modify the generating curve and not the surface itself. If you reshape the surface, its link to the generating curve will be broken and the surface will lose associativity and become a generic surface.
Deleting the Curves that Generate the Solid or Surface

The DELOBJ system variable controls whether the curves that generate an object are automatically deleted after the solid or surface is created. However, if surface associativity is on, the DELOBJ setting is ignored and the generating curves are not deleted.

Create a Solid or Surface by Extruding

Create a 3D solid or surface by stretching curves into 3D space.

The EXTRUDE command creates a solid or surface that extends the shape of a curve. Open curves create surfaces and closed curves create solids or surfaces. See Overview of Creating Solids and Surfaces on page 446.

Options for Extrusion

When you extrude objects, you can specify any of the following options:

■ **Mode.** Sets whether the extrude creates a surface or a solid.

■ **Specify a path for extrusion.** With the Path option, create a solid or surface by specifying an object to be the path for the profile, or shape, of the extrusion. The extruded object starts from the plane of the profile and ends on a plane perpendicular to the path at the endpoint of the path. For best results, use object snaps to make sure that the path is on or within the boundary of the object being extruded.
Extruding is different from sweeping. When you extrude a profile along a path, the profile follows the shape of the path, even if the path does not intersect the profile. Sweeping usually provides greater control and better results.

- **Taper angle.** Tapering the extrusion is useful for defining part that require a specific taper angle, such as a mold used to create metal products in a foundry.

- **Direction.** With the Direction option, you can specify two points to set the length and direction of the extrusion.

- **Expression.** Enter a mathematical expression to constrain the height of the extrusion. See *Create Geometric Relationships between Associative Surfaces* on page 508.

**Quick Reference**

**Commands**

3DOSNAP

Sets the object snap modes for 3D objects.
BREP
Removes the history from 3D solids and composite solids, and associativity from surfaces.

EXTRUDE
Creates a 3D solid or surface by extending the dimensions of an object.

JOIN
Joins the endpoints of lines, 2D and 3D polylines, arcs, elliptical arcs, helixes, and splines to create single object.

LIST
Displays property data for selected objects.

MEASUREGEOM
Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

REGION
Converts an object that encloses an area into a region object.

System Variables

3DOSMODE
Controls the settings for the 3D object snaps.

DELOBJ
Controls whether geometry used to create 3D objects is retained or deleted.

SOLIDHIST
Controls the default history property setting for solid objects.

SUBOBJSELECTIONMODE
Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

SURFACEASSOCIATIVITY
Controls whether surfaces maintain a relationship with the objects from which they were created.

SURFACEASSOCIATIVITYDRAG
Sets the dragging preview behavior of associative surfaces.
SURFACEMODELINGMODE

Controls whether surfaces are created as procedural surfaces or NURBS surfaces.

Create a Solid or Surface by Sweeping

Create a 3D solid or surface by sweeping a profile along a path.

The SWEEP command creates a solid or surface by extending a profile shape (the object that is swept) along a specified path. When you sweep a profile along a path, the profile is moved and aligned normal (perpendicular) to the path. Open profiles create surfaces and closed curves create solids or surfaces. See Overview of Creating Solids and Surfaces on page 446.

You can sweep more than one profile object along a path.

Options for Sweeping

When you extrude objects, you can specify any of the following options:

- **Mode**. Sets whether the sweep creates a surface or a solid.
- **Alignment**. If the profile is not on the same plane as the sweep path, specify how the profile aligns with the sweep path.
- **Base Point**. Specify a base point on the profile to sweep along the profile.
- **Scale**. Specify a value that will change the size of the object from the beginning of the sweep to the end. Enter a mathematical expression to constrain the object scaling. See Create Geometric Relationships between Associative Surfaces on page 508.
Twist. By entering a twist angle, the object rotates along the length of the profile. Enter a mathematical expression to constrain the object's twist angle. See Create Geometric Relationships between Associative Surfaces on page 508.

See also:
- Create Associative Surfaces on page 505
- Create Geometric Relationships between Associative Surfaces on page 508
- Draw Splines on page 281
- Modify Splines on page 359
Quick Reference

Commands

3DOSNAP
Sets the object snap modes for 3D objects.

BREP
Removes the history from 3D solids and composite solids, and associativity from surfaces.

JOIN
Joins the endpoints of lines, 2D and 3D polylines, arcs, elliptical arcs, helixes, and splines to create single object.

LIST
Displays property data for selected objects.

MEASUREGEOM
Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

REGION
Converts an object that encloses an area into a region object.

SWEEP
Creates a 3D solid or surface by sweeping a 2D or 3D object or subobject along a path.

System Variables

3DOSMODE
Controls the settings for the 3D object snaps.

DELOBJ
Controls whether geometry used to create 3D objects is retained or deleted.

SOLIDHIST
Controls the default history property setting for solid objects.
SUBOBJSELECTIONMODE
Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

SURFACEASSOCIATIVITY
Controls whether surfaces maintain a relationship with the objects from which they were created.

SURFACEASSOCIATIVITYDRAG
Sets the dragging preview behavior of associative surfaces.

SURFACEMODELINGMODE
Controls whether surfaces are created as procedural surfaces or NURBS surfaces.

Create a Solid or Surface by Lofting
Create a 3D solid or surface by lofting a profile through a set of two or more cross-section profiles.

The cross-section profiles define the shape of the resulting solid object.

Cross-section profiles can be open or closed curves. Open curves create surfaces and closed curves create solids or surfaces. See Overview of Creating Solids and Surfaces on page 446.

Options for Lofting
- **Mode.** Sets whether the loft creates a surface or a solid.
- **Cross-section profiles.** Select a series of cross-section profiles to define the shape of the new 3D object.
lofted objects with different cross-section settings

As you create a lofted object, you can adjust its shape by specifying how the profile passes through the cross sections (for example, a sharp or smooth curve). You can also modify the settings later in the Properties Inspector. For more information, see Modify Properties of 3D Solid, Surface, and Mesh on page 590.

- **Paths.** Specify a path for the loft operation to obtain more control over the shape of the lofted object. For best results, start the path curve on the plane of the first cross section and end it on the plane of the last cross section.

- **Guide curves.** Specify guide curves to match points on corresponding cross sections. This method prevents undesired results, such as wrinkles in the resulting 3D object.
Each guide curve must meet the following criteria:

- Intersects each cross section
- Starts on the first cross section
- Ends on the last cross section

See also:

- Create Associative Surfaces on page 505
- Create Geometric Relationships between Associative Surfaces on page 508
- Draw Splines on page 281
- Modify Splines on page 359

Quick Reference

Commands

3DOSNAP
Sets the object snap modes for 3D objects.

BREP
Removes the history from 3D solids and composite solids, and associativity from surfaces.

JOIN
Joins the endpoints of lines, 2D and 3D polylines, arcs, elliptical arcs, helixes, and splines to create single object.

LIST
Displays property data for selected objects.

MEASUREGEOM
Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

LOFT
Creates a 3D solid or surface in the space between several cross sections.
REGION
Converts an object that encloses an area into a region object.

**System Variables**

*3DOSMODE*
Controls the settings for the 3D object snaps.

*DELOBJ*
Controls whether geometry used to create 3D objects is retained or deleted.

*SOLIDHIST*
Controls the default history property setting for solid objects.

*SUBOBJSELECTIONMODE*
Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

*SURFACEASSOCIATIVITY*
Controls whether surfaces maintain a relationship with the objects from which they were created.

*SURFACEASSOCIATIVITYDRAG*
Sets the dragging preview behavior of associative surfaces.

*SURFACEMODELINGMODE*
Controls whether surfaces are created as procedural surfaces or NURBS surfaces.

---

**Create a Solid or Surface by Revolving**

Create a 3D object by revolving curves about an axis.

When the Mode option is set to Surface, you will create a surface and if Mode is set to Solid you will create a solid regardless of whether the curve is open or closed. See [Overview of Creating Solids and Surfaces](#) on page 446 for more information. When revolving a solid, you can only use a revolve angle of 360 degrees.

**Options for Revolving**

- **Mode.** Sets whether the revolve creates a surface or a solid.
■ **Start Angle.** Specifies an offset for the revolution from the plane of the object being revolved.

■ **Reverse.** Changes the direction of the revolve.

■ **Expression.** Enter a formula or equation to specify the revolve angle. This option is only available if you are creating associative surfaces. See [Create Geometric Relationships between Associative Surfaces](#) on page 508.

**See also:**

■ [Create Associative Surfaces](#) on page 505

■ [Draw Splines](#) on page 281

■ [Modify Splines](#) on page 359

**Quick Reference**

**Commands**

3DOSNAP

Sets the object snap modes for 3D objects.

BREP

Removes the history from 3D solids and composite solids, and associativity from surfaces.

JOIN

Joins the endpoints of lines, 2D and 3D polylines, arcs, elliptical arcs, helixes, and splines to create single object.

LIST

Displays property data for selected objects.

MEASUREGEOM

Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

REGION

Converts an object that encloses an area into a region object.
REVOLVE

Creates a 3D solid or surface by sweeping an object around an axis.

**System Variables**

3DOSMODE

Controls the settings for the 3D object snaps.

DELOBJ

Controls whether geometry used to create 3D objects is retained or deleted.

SOLIDHIST

Controls the default history property setting for solid objects.

SUBOBJSELECTIONMODE

Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

SURFACEASSOCIATIVITY

Controls whether surfaces maintain a relationship with the objects from which they were created.

SURFACEASSOCIATIVITYDRAG

Sets the dragging preview behavior of associative surfaces.

SURFACEMODELINGMODE

Controls whether surfaces are created as procedural surfaces or NURBS surfaces.

**Create Solids**

Create 3D solids from primitives or by combining or extending existing objects.

**Overview of Creating 3D Solids**

3D solid objects often start as one of several basic shapes, or primitives, that you can then modify and recombine. A 3D solid can also be the result of extruding a 2D shape to follow a specified path in 3D space.
About Solid Primitives

You can create several basic 3D shapes, known as *solid primitives*: boxes, cones, cylinders, spheres, wedges, pyramids, and tori (donuts).

By combining primitive shapes, you can create more complex solids. For example, you can join two solids, subtract one from the other, or create a shape based on the intersection of their volumes.

About Solids Based on Other Objects

You can also create 3D solids from 2D geometry or other 3D objects.

The following methods are available:

- **Sweep**. Extends a 2D object along a path.
- **Extrusion**. Extends the shape of a 2D object in a perpendicular direction into 3D space.
■ **Revolve.** Sweeps a 2D object around an axis.

■ **Loft.** Extends the contours of a shape between one or more open or closed objects.

■ **Slice.** Divides a solid object into two separate 3D objects.

■ **Sculpting Surfaces.** Converts and trims a group of surfaces that enclose a watertight area into a solid.

■ **Conversion.** Converts mesh objects and planar objects with thickness into solids and surfaces.

**Quick Reference**

**Commands**

**BOX**

Creates a 3D solid box.

**CONE**

Creates a 3D solid cone.

**CONVTSOLID**

Converts 3D meshes and polylines and circles with thickness to 3D solids.

**CONVTOSURFACE**

Converts objects to 3D surfaces.

**CYLINDER**

Creates a 3D solid cylinder.

**EXTRUDE**

Creates a 3D solid or surface by extending the dimensions of an object.

**INTERSECT**

Creates a 3D solid, surface, or 2D region from overlapping solids, surfaces, or regions.

**INSERT**

Inserts a block or drawing into the current drawing.
LOFT
Creates a 3D solid or surface in the space between several cross sections.

PYRAMID
Creates a 3D solid pyramid.

REVOLVE
Creates a 3D solid or surface by sweeping an object around an axis.

SLICE
Creates new 3D solids and surfaces by slicing, or dividing, existing objects.

SUBTRACT
Combines selected 3D solids or 2D regions by subtraction.

SURFSCULPT
Trims and combines surfaces that bound a watertight area to create a solid.

SWEEP
Creates a 3D solid or surface by sweeping a 2D or 3D object or subobject along a path.

TORUS
Creates a donut-shaped 3D solid.

UNION
Combines selected 3D solids, surfaces, or 2D regions by addition.

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

WEDGE
Creates a 3D solid wedge.

**System Variables**

SURFACEMODELINGMODE
Controls whether surfaces are created as procedural surfaces or NURBS surfaces.
FACETRES
Adjusts the smoothness of shaded and rendered objects and objects with hidden lines removed.

ISOLINES
Specifies the number of contour lines per surface on objects.

**Create 3D Solid Primitives**
Start with standard shapes known as *solid primitives* to create boxes, cones, cylinders, spheres, tori (donuts), wedges, and pyramids.

**Create a Solid Box**
Create a rectangular or cubical solid box.

The base of the box is always drawn parallel to the XY plane of the current UCS (workplane).

**Box Creation Options**
Use the following options to control the size and rotation of the boxes you create:

- **Create a cube.** Use the Cube option of the BOX command to create a box with sides of equal length.

- **Specify rotation.** Use the Cube or Length option if you want to set the rotation of the box in the XY plane.
■ Start from the center point. Use the Center Point option to create a box using a specified center point.

Quick Reference

Commands
BOX

Creates a 3D solid box.

System Variables
DRAGVS

Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Create a Solid Wedge

Create a solid wedge with rectangular or cubical faces.

The base of the wedge is drawn parallel to the XY plane of the current UCS with the sloped face opposite the first corner. The height of the wedge is parallel to the Z axis.
Wedge Creation Options

Use the following options to control the size and rotation of the wedges you create:

 ■ **Create a wedge with sides of equal length.** Use the Cube option of the WEDGE command.

 ■ **Specify rotation.** Use the Cube or Length option if you want to set the rotation of the wedge in the XY plane.

 ■ **Start from the center point.** Use the Center Point option to create a wedge using a specified center point.

Quick Reference

Commands

**WEDGE**

Creates a 3D solid wedge.

System Variables

**DRAGVS**

Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Create a Solid Cone

Create a pointed or frustum of a cone with a circular or elliptical base.
By default, the base of the cone lies on the $XY$ plane of the current UCS. The height of the cone is parallel to the $Z$ axis.

**Cone Creation Options**

Use the following options to control the size and rotation of the cones you create:

- **Set the height and orientation.** Use the Axis Endpoint option of the CONE command. Use the Top Radius option to specify the axis endpoint as the point of the cone or the center of the top face. The axis endpoint can be located anywhere in 3D space.

- **Create a frustum of a cone.** Use the Top Radius option of the CONE command to create a frustum, which tapers to an elliptical or planar face.

The Frustum tool is also available from the Modeling tab of the tool palette. You can also use grips to modify the tip of a cone and convert it to a flat face.

- **Specify circumference and base plane.** The 3P (Three Points) option of the CONE command defines the size and plane of the base of the cone anywhere in 3D space.

- **Define the angle of the taper.** To create a conical solid that requires a specific angle to define its sides, draw a 2D circle. Then use EXTRUDE and the Taper Angle option to taper the circle at an angle along the $Z$ axis. This method, however, creates an extruded solid, not a true solid cone primitive.
See also:

- Use Grips to Edit 3D Solids and Surfaces on page 567

Quick Reference

Commands

CONE

Creates a 3D solid cone.

System Variables

DRAGVS

Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Create a Solid Cylinder

Create a solid cylinder with a circular or elliptical base.

By default, the base of the cylinder lies on the XY plane of the current UCS. The height of the cylinder is parallel to the Z axis.
Cylinder Creation Options

Use the following options to control the size and rotation of the cylinders you create:

- **Set rotation.** Use the Axis Endpoint option of the CYLINDER command to set the height and rotation of the cylinder. The center point of the top plane of the cylinder is the axis endpoint, which can be located anywhere in 3D space.

- **Use three points to define the base.** Use the 3P (Three Points) option to define the base of the cylinder. You can set three points anywhere in 3D space.

- **Construct a cylindrical form with special detail, such as grooves.** Create a closed polyline (PLINE) to represent a 2D profile of the base. Use EXTRUDE to define the height along the Z axis. The resulting extruded solid is not a true solid cylinder primitive.

Quick Reference

Commands

CYLINDER

Creates a 3D solid cylinder.

System Variables

DRAGVS

Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Create a Solid Sphere

Create a solid sphere using one of several methods.
When you start with the center point, the central axis of the sphere parallels the Z axis of the current user coordinate system (UCS).

**Sphere Creation Options**

Use the following options to draw a sphere with the SPHERE command:

- **Specify three points to set the size and plane of the circumference or radius.** Use the 3P (Three Points) option to define the size of the sphere anywhere in 3D space. The three points also define the plane of the circumference.

- **Specify two points to set the circumference or radius.** Use the 2P (Two Points) option to define the size of the sphere anywhere in 3D space. The plane of the circumference matches the Z value of the first point.

- **Set the size and location of the sphere based on other objects.** Use the Ttr (Tangent, Tangent, Radius) option to define a sphere that is tangent to two circles, arcs, lines, and some 3D objects. The tangency points are projected onto the current UCS.

**Quick Reference**

**Commands**

SPHERE

Creates a 3D solid sphere.

**System Variables**

DRAGVS

Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.
Create a Solid Pyramid

Create a solid pyramid with up to 32 sides.

You can create a pyramid that tapers to a point, or create a frustum of a pyramid, which tapers to a planar face.

Pyramid Creation Options

Use the following options to control the size, shape, and rotation of the pyramids you create:

- **Set the number of sides.** Use the Sides option of the PYRAMID command to set the number of sides for the pyramid.

- **Set the length of the edges.** Use the Edges option to specify the dimension of the sides at the base.

- **Create a frustum of a pyramid.** Use the Top Radius option to create a frustum, which tapers to a planar face. The frustum face is parallel to, and has the same number of sides as, the base.
Set the height and rotation of the pyramid. Use the Axis Endpoint option of the PYRAMID command to specify the height and rotation of the pyramid. This endpoint, or top of the pyramid, can be located anywhere in 3D space.

Quick Reference

Commands
PYRAMID
Creates a 3D solid pyramid.

System Variables
DRAGVS
Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Create a Solid Torus
Create a ring-shaped solid that resembles the inner tube of a tire.
A torus has two radius values. One value defines the tube. The other value defines the distance from the center of the torus to the center of the tube. By default, a torus is drawn parallel to and is bisected by the $XY$ plane of the current UCS.

A torus can be self-intersecting. A self-intersecting torus has no center hole because the radius of the tube is greater than the radius of the torus.

Torus Creation Options

Use the following options to control the size and rotation of the tori you create.

- **Set the size and plane of the circumference or radius.** Use the 3P (Three Points) option to define the size of the torus anywhere in 3D space. The three points also define the plane of the circumference. Use this option to rotate the torus as you create it.

- **Set the circumference or radius.** Use the 2P (Two Points) option to define the size of the torus anywhere in 3D space. The plane of the circumference matches the $Z$ value of the first point.

- **Set the size and location of the torus based on other objects.** Use the Ttr (Tangent, Tangent, Radius) option to define a torus that is tangent to two circles, arcs, lines, and some 3D objects. The tangency points are projected onto the current UCS.
Quick Reference

Commands

TORUS

Creates a donut-shaped 3D solid.

System Variables

DRAGVS

Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Create a Polysolid

Use the same techniques you use to create polylines to create a polysolid object.

The POLYSOLID command provides a quick way to draw 3D walls. A polysolid is like an extruded, wide polyline. In fact, you can draw polysolids the same way that you draw a polyline, using both straight and curved segments. Unlike extruded polylines, which lose any width properties upon extrusion, polysolids retain the width of their line segments.

You can also convert objects such as a line, 2D polyline, arc, or circle to a polysolid.

Polysolids are displayed as swept solids in the Properties Inspector.
Polysolid Creation Options

Use the following options to control the size and shape of the polysolids you create:

- **Create arced segments.** Use the Arc option to add curved segments to the polysolid. The profile of a polysolid with curved segments remains perpendicular to the path.

- **Create a polysolid from a 2D object.** Use the Object option to convert an object such as a polyline, circle, line, or arc to a polysolid. The DELOBJ system variable controls whether the path (a 2D object) is automatically deleted when you create a polysolid.

- **Close the gap between the first and last points.** Use the Close option to create a connecting segment.

- **Set the height and width.** Use the Height and Width options for the POLYSOLID command. The values you set are stored in the PSOLWIDTH and PSOLHEIGHT system variables.

- **Set where the object is drawn in relation to the specified points.** Use the Justification option to place the path of the polysolid to the right, to the left, or down the center of the points you specify.

Quick Reference

Commands

POLYSOLID

Creates a 3D wall-like polysolid.
System Variables

DELOBJ
Controls whether geometry used to create 3D objects is retained or deleted.

PSOLHEIGHT
Controls the default height for a swept solid object created with the POLYSOLID command.

PSOLWIDTH
Controls the default width for a swept solid object created with the POLYSOLID command.

Create 3D Solids from Objects

Convert existing objects to 3D solids.

You can use several methods to convert objects in your drawing to 3D solids:

- Convert surfaces and objects with Thickness to 3D Solids
- Convert a group of surfaces to a 3D solid
- Convert mesh to 3D solids
- Thicken surfaces to convert them to 3D solids

mesh and polyline with thickness converted to optimized 3D solids

The DELOBJ system variable controls whether the objects you select are automatically deleted when the 3D object is created.
Convert Surfaces and Objects with Thickness to 3D Solids

You can convert different types of objects into extruded 3D solids with the CONVTOSOLID command. These objects include closed polylines and circles with thickness, as well as watertight meshes and surfaces. For a complete list of objects that can be converted using this method, see CONVTOSOLID.

Convert a Group of Surfaces to a 3D Solid

Use the SURFSCULPT command to convert a group of surfaces that enclose a watertight region to a 3D solid.

Convert Mesh to 3D Solids

When you convert mesh objects to 3D solids, the shape of the new solid object approximates, but does not exactly duplicate, the original mesh object. You can control the differentiation somewhat by specifying whether the result is smooth or faceted (SMOOTHMESHCONVERT). You can also specify whether the resulting faces are merged (optimized).

For example, if you convert a mesh box to a solid object, following options are available:

- **Smoothed and optimized.** Coplanar faces are merged into a single face. The overall shape of some faces can change. Edges of faces that are not coplanar are rounded. (SMOOTHMESHCONVERT = 0)

![Smoothed and optimized example](image)

- **Smoothed and not optimized.** Each original mesh face is retained in the converted object. Edges of faces that are not coplanar are rounded. (SMOOTHMESHCONVERT = 1)
■ **Faceted and optimized.** Coplanar faces are merged into a single, flat face. The overall shape of some faces can change. Edges of faces that are not coplanar are creased, or angular. (SMOOTHMESHCONVERT = 2)

■ **Faceted and not optimized.** Each original mesh face is converted to a flat face. Edges of faces that are not coplanar are creased, or angular. (SMOOTHMESHCONVERT = 3)

You cannot convert the following types of mesh objects to a 3D solid:

■ **Mesh with gaps between faces.** Gizmo editing can sometimes result in gaps, or holes between the faces. In some cases, you can close the gaps by smoothing the mesh object.

■ **Mesh that has self-intersecting boundaries.** If you have modified a mesh object so that one or more faces intersect faces in the same object, you cannot convert it to a 3D solid.
In some cases, mesh that is not eligible to be converted to a solid object can be converted to a surface.

**Thicken Surfaces to Convert Them to 3D Solids**

You can convert 3D surface objects to 3D solids with the THICKEN command.

Grip editing is limited for objects that are created using this method.

**Quick Reference**

**Commands**

CONVTOSOLID
- Converts 3D meshes and polylines and circles with thickness to 3D solids.

SURFSCULPT
- Trims and combines surfaces that bound a watertight area to create a solid.

THICKEN
- Converts a surface into a 3D solid with a specified thickness.

UNION
- Combines selected 3D solids, surfaces, or 2D regions by addition.

**System Variables**

DELOBJ
- Controls whether geometry used to create 3D objects is retained or deleted.
SMOOTHMESHCONVERT

Sets whether mesh objects that you convert to 3D solids or surfaces are smoothed or faceted, and whether their faces are merged.

**Combine or Slice 3D Objects**
Create new composite 3D objects or slice objects to divide them.

**Create Composite Objects**
Create composite 3D objects by combining, subtracting, or finding the intersecting mass of two or more 3D solids, surfaces, or regions.

Composite solids are created from two or more solids, surfaces, or regions through any of the following commands: UNION, SUBTRACT, and INTERSECT.

3D solids record a history of how they were created. This history allows you to see the original forms that make up composite solids. For more information, see Display Original Forms of Composite Solids on page 582.

**Methods for Creating Composite Objects**
Three methods are available for creating composite solids, surfaces, or regions:

- **Combine two or more objects.**
  With UNION, you can combine the total volume of two or more objects.

  ![Diagrams](image)

  objects to be combined
  
  result

- **Subtract one set of solids from another.**
  With SUBTRACT, you can remove the common area of one set of solids from another. For example, you can use SUBTRACT to add holes to a mechanical part by subtracting cylinders from the object.
Find the common volume.
With INTERSECT, you can create a composite solid from the common volume of two or more overlapping solids. INTERSECT removes the portions that do not overlap and creates a composite solid from the common volume.

Create Composites from Mixed Object Types
In addition to creating composite objects from the same object types, you can also create composites from mixed surfaces and solids.

- Mixed intersections. Combining a solid and a surface through intersection results in a surface.

- Mixed subtractions. Subtracting a 3D solid from a surface results in a surface. However, you cannot subtract a surface from a 3D solid object.

- Mixed unions. You cannot create a union between 3D solid and surface objects.

You cannot combine solids with mesh objects. However, you can convert them to 3D solids in order to combine them with solids.

If a selection set of mixed objects contains regions, the regions are ignored.
Quick Reference

Commands

INTERSECT
creates a 3D solid, surface, or 2D region from overlapping solids, surfaces, or regions.

SUBTRACT
combines selected 3D solids or 2D regions by subtraction.

UNION
combines selected 3D solids, surfaces, or 2D regions by addition.

System Variables

SHOWHIST
controls the Show History property for solids in a drawing.

SOLIDHIST
controls the default history property setting for solid objects.

Create 3D Solids by Slicing

Create new 3D solids by slicing, or dividing, existing objects.

When you use the SLICE command to slice a 3D solid, you can define the cutting plane in several ways. For example, you can specify three points, an axis, a surface, or a planar object to act as a cutting plane. You can retain one or both halves of the sliced object.

Sliced 3D solids do not retain a history of the original forms that created them. However, they do retain the layer and color properties of the original objects.
For a complete list of objects that can be used for a slice operation, see SLICE.

See also:
- Create Sections and 2D Drawings from 3D Models on page 635

## Quick Reference

### Commands

SLICE

Creates new 3D solids and surfaces by slicing, or dividing, existing objects.

---

### Check 3D Models for Interferences

Find areas where 3D solids or surfaces intersect or overlap.

Use the INTERFERE command to check for areas of interference within a set of 3D solid or surface models. You can compare two sets of objects or check all 3D solids and surfaces in a drawing.

Interference checking creates temporary solid or surface objects and highlights where the models intersect.

If the selection set contains both 3D solids and surfaces, the resulting interference object is a surface.

You cannot check interference for mesh objects. However, if you select mesh objects, you can choose to convert them to a solid or surface object and continue the operation.

During the checking operation, you can use the Interference Checking dialog box to cycle through and zoom to interference objects. You can also specify whether to delete the temporary objects that are created during interference checking.
Methods for Checking Interference

You can check interference using the following methods:

- **Define one selection set.** Check the interference of all the 3D solids and surfaces in a single selection set.
- **Define two selection sets.** Check the interference of the objects in the first set of objects against the objects in the second selection set.
- **Individually specify solids that are nested within blocks or xrefs.** Individually select 3D solid or surface objects that are nested in blocks and external references (xrefs) and compare them against other objects in the selection set.

Quick Reference

Commands

**INTERFERE**

Creates a temporary 3D solid from the interferences between two sets of selected 3D solids.

System Variables

**INTERFERECOLOR**

Sets the color for interference objects.

**INTERFEREOBJVS**

Sets the visual style for interference objects.

**INTERFEREVPVS**

Specifies the visual style for the viewport during interference checking.

Create Surfaces

Create procedural surfaces and NURBS surfaces from curves or by combining or extending existing surfaces.
Overview of Creating Surfaces

Surface modeling provides the ability to edit multiple surfaces as an associative group or in a more free-form way.

In addition to 3D solid and mesh objects, AutoCAD for Mac offer two type of surfaces: procedural and NURBS.

- **Procedural Surfaces** can be associative, maintaining relationships with other objects so that they can be manipulated as a group.

- **NURBS Surfaces** - are not associative. Instead, they have control vertices that allow you to sculpt shapes in a more natural way.

Use procedural surfaces to take advantage of associative modeling, and use NURBS surfaces to take advantage of sculpting with control vertices. The illustration below shows a procedural surface on the left, and a NURBS surface on the right.

Choose a Surface Creation Method

Create procedural and NURBS surfaces using the following methods:

- **Create surfaces from profiles** on page 446. Create surfaces from profile shapes composed of lines and curves with EXTRUDE, LOFT, PLANESURF, REVOLVE, SURFNETWORK, and SWEEP.
- **Create surfaces from other surfaces** on page 492. Blend, patch, extend, fillet, and offset surfaces to create new surfaces (SURFBLND, SURFPATCH, SURFEXTEND, SURFFILLET and SURFOFFSET).

- **Convert objects into procedural surfaces** on page 501. Convert existing solids (including composite objects), surfaces, and meshes into procedural surfaces (CONVTOSURFACE command).

- **Convert procedural surfaces into NURBS surfaces** on page 501. Some objects cannot be converted directly to NURBS (for example, mesh objects). In that case, convert the object to a procedural surface and then convert it to a NURBS surface. (CONVTONURBS command).
Understand Surface Continuity and Bulge Magnitude

Surface continuity and bulge magnitude are properties that are frequently used when creating surfaces. When you create a new surface, you can specify the continuity and bulge magnitude with special grips.

Continuity is a measure of how smoothly two curves or surfaces flow into each other. The type of continuity can be important if you need to export your surfaces to other applications.
Continuity types include the following:

■ **G0 (Position)**. Measures location only. If the edge of each surface is collinear, the surfaces are positionally continuous (G0) at the edge curves. Note that two surfaces can meet at any angle and still have positional continuity.

■ **G1 (Tangency)**. Includes both positional and tangential continuity (G0 + G1). With tangentially continuous surfaces, the end tangents match at the common edges. The two surfaces appear to be traveling in the same direction at the join, but they may have very different apparent “speeds” (or rates of change in direction, also called curvature).

■ **G2 (Curvature)**. Includes positional, tangential, and curvature continuity (G0 + G1+G2). The two surfaces share the same curvature.

Bulge magnitude is a measure of how much surface curves or “bulges” as it flows into another surface. Magnitude can be between 0 and 1 where 0 is flat and 1 curves the most.

**Set Surface Properties Before and After Creation**

Set defaults that control a variety of surface properties before and after you create the surface objects.

■ Surface Modeling System Variables. There are a number of system variables that are frequently used and changed during surface creation: SURFACEMODELINGMODE, SURFACEASSOCIATIVITY, SURFACEASSOCIATIVITYDRAG, SURFACEAUTOTRIM, and SUBOBJSELECTIONMODE.

■ Properties Inspector. Modifies properties for both the surface objects and their subobjects after they are created. For example, you can change the number of isolines in the U and V directions.

**Quick Reference**

**Commands**

3DOSNAP

Sets the object snap modes for 3D objects.

ANALYSISZEBRA

Projects stripes onto a 3D model to analyze surface continuity.
BREP
Removes the history from 3D solids and composite solids, and associativity from surfaces.

CONVTONURBS
Converts 3D solids and surfaces into NURBS surfaces.

CONVTOSURFACE
Converts objects to 3D surfaces.

EXTRUDE
Creates a 3D solid or surface by extending the dimensions of an object.

JOIN
Joins the endpoints of lines, 2D and 3D polylines, arcs, elliptical arcs, helixes, and splines to create single object.

LOFT
Creates a 3D solid or surface in the space between several cross sections.

PLANESURF
Creates a planar surface.

REVOLVE
Creates a 3D solid or surface by sweeping an object around an axis.

SURFBLEND
Creates a continuous blend surface between two existing surfaces.

SURFNETWORK
Creates a surface in the space between several curves in the U and V directions (including surface and solid edge subobjects).

SURFOFFSET
Creates a parallel surface a specified distance from the original surface.

SURFPATCH
Creates a new surface by fitting a cap over a surface edge that forms a closed loop.
SWEEP

Creates a 3D solid or surface by sweeping a 2D or 3D object or subobject along a path.

VISUALSTYLES

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

System Variables

3DOSMODE

Controls the settings for the 3D object snaps.

DELOBJ

Controls whether geometry used to create 3D objects is retained or deleted.

FACETRES

Adjusts the smoothness of shaded and rendered objects and objects with hidden lines removed.

ISOLINES

Specifies the number of contour lines per surface on objects.

SUBOBJSELECTIONMODE

Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

SURFACEASSOCIATIVITY

Controls whether surfaces maintain a relationship with the objects from which they were created.

SURFACEASSOCIATIVITYDRAG

Sets the dragging preview behavior of associative surfaces.

SURFACEMODELINGMODE

Controls whether surfaces are created as procedural surfaces or NURBS surfaces.

Create Procedural Surfaces

Create procedural surfaces by blending, patching, and offsetting existing surfaces or by converting 3D solids, meshes and other planar geometry.
Create Surfaces from Other Surfaces

There are many ways to create procedural surfaces from existing surfaces. These include blending, patching, and offsetting or creating network and planar surfaces.

When you create procedural surfaces, use SURFACEASSOCIATIVITY to work with the surfaces as a group. Just as you can modify the face of a solid and have the entire solid update, when you modify a group of surfaces that are associative, all of the surfaces update accordingly.

Create Planar Surfaces

Create planar surfaces in the space between edge subobjects, splines and other 2D and 3D curves.

Create planar surfaces with the PLANESURF command. Planar surfaces can be created from multiple closed objects and the curves can be surface or solid edge subobjects. During creation, specify the tangency and bulge magnitude.

See also:

- Overview of Creating Surfaces on page 486

Quick Reference

Commands

3DOSNAP
- Sets the object snap modes for 3D objects.

LIST
- Displays property data for selected objects.

MEASUREGEOM
- Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

PLANESURF
- Creates a planar surface.

UCS
- Manages user coordinate systems.
VISUALSTYLES

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

System Variables
3DOSMODE

Controls the settings for the 3D object snaps.

DELOBJ

Controls whether geometry used to create 3D objects is retained or deleted.

LIST

Displays property data for selected objects.

MEASUREGEOM

Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

SUBOBJSELECTIONMODE

Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

SURFACEASSOCIATIVITY

Controls whether surfaces maintain a relationship with the objects from which they were created.

Create Network Surfaces

Create non-planar surfaces in the space between edge subobjects, splines and other 2D and 3D curves.

Create non-planar, network surfaces with the SURFNETWORK command. Network surfaces are similar to lofted surfaces in that they are created in the space between several curves in the U and V directions. The curves can be surface or solid edge subobjects. When you create the surface you can specify the tangency and bulge magnitude of the surface edges.

See also:

■ Overview of Creating Surfaces on page 486
Quick Reference

Commands

3DOSNAP
Sets the object snap modes for 3D objects.

LIST
Displays property data for selected objects.

MEASUREGEOM
Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

SURFNETWORK
Creates a surface in the space between several curves in the U and V directions (including surface and solid edge subobjects).

UCS
Manages user coordinate systems.

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

System Variables

3DOSMODE
Controls the settings for the 3D object snaps.

DELOBJ
Controls whether geometry used to create 3D objects is retained or deleted.

LIST
Displays property data for selected objects.

MEASUREGEOM
Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

SUBOBJSELECTIONMODE
Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.
SURFACEASSOCIATIVITY

Controls whether surfaces maintain a relationship with the objects from which they were created.

Blend a Surface

Create a transition surface between two existing surfaces.

Use SURFBLEND to create a new surface between existing surfaces and solids. When you blend surfaces together, specify the surface continuity and bulge magnitude for the start and end edges.

See also:
- Overview of Creating Surfaces on page 486

Quick Reference

Commands

3DOSNAP
- Sets the object snap modes for 3D objects.
LIST
- Displays property data for selected objects.
MEASUREGEOM
- Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.
SURFBLEND
- Creates a continuous blend surface between two existing surfaces.
UCS
- Manages user coordinate systems.
VISUALSTYLES
- Creates and modifies visual styles and applies a visual style to a viewport.
**System Variables**

3DOSMODE
- Controls the settings for the 3D object snaps.

CULLINGOBJ
- Controls whether 3D subobjects that are hidden from view can be highlighted or selected.

CULLINGOBJSELECTION
- Controls whether 3D objects that are hidden from view can be highlighted or selected.

DELOBJ
- Controls whether geometry used to create 3D objects is retained or deleted.

SUBOBJSELECTIONMODE
- Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

SURFACEASSOCIATIVITY
- Controls whether surfaces maintain a relationship with the objects from which they were created.

**Patch a Surface**

Create a surface by patching a closed surface or curve.

Use SURFPATCH to create a surface inside a closed curve (such as a closed spline) that is another surface’s edge. You can also draw a guide curve to constrain the shape of the patch surface with the constrain geometry option. When you patch surfaces, specify the continuity and bulge magnitude.

**See also:**

- [Overview of Creating Surfaces](#) on page 486
Quick Reference

Commands

3DOSNAP
Sets the object snap modes for 3D objects.

LIST
Displays property data for selected objects.

MEASUREGEOM
Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

SURFPATCH
Creates a new surface by fitting a cap over a surface edge that forms a closed loop.

UCS
Manages user coordinate systems.

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

System Variables

3DOSMODE
Controls the settings for the 3D object snaps.

CULLINGOBJ
Controls whether 3D subobjects that are hidden from view can be highlighted or selected.

CULLINGOBJSELECTION
Controls whether 3D objects that are hidden from view can be highlighted or selected.

DELOBJ
Controls whether geometry used to create 3D objects is retained or deleted.
SUBOBJSELECTIONMODE

Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

SURFACEASSOCIATIVITY

Controls whether surfaces maintain a relationship with the objects from which they were created.

Offset a Surface

Create a parallel surface a specified distance from the original surface.

With SURFOFFSET specify the offset distance and whether or not the offset surface maintains associativity with the original surface. You can also specify the offset distance with a mathematical expression. See Constrain a Design with Formulas and Equations on page 398.

Surface Offset Options

When you offset a surface, you can do the following:

- Change the offset direction with the Flip option

- Offset in both directions to create two new surfaces
■ Create a solid between the offset surfaces

■ If you are offsetting more than one surface, you can specify whether the offset surfaces remain connected.

■ Enter an expression that will constrain the distance of the offset surface to the original surface. This option only appears if associativity is on. See Constrain a Design with Formulas and Equations on page 398.

See also:
■ Overview of Creating Surfaces on page 486

Quick Reference

Commands

3DOSNAP

Sets the object snap modes for 3D objects.
LIST
Displays property data for selected objects.

MEASUREGEOM
Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

SURFOFFSET
Creates a parallel surface a specified distance from the original surface.

UCS
Manages user coordinate systems.

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

System Variables

3DOSMODE
Controls the settings for the 3D object snaps.

CULLINGOBJ
Controls whether 3D subobjects that are hidden from view can be highlighted or selected.

CULLINGOBJSELECTION
Controls whether 3D objects that are hidden from view can be highlighted or selected.

DELOBJ
Controls whether geometry used to create 3D objects is retained or deleted.

SUBOBJSELECTIONMODE
Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

SURFACEASSOCIATIVITY
Controls whether surfaces maintain a relationship with the objects from which they were created.
Convert Objects to Procedural Surfaces

Convert 3D solids, meshes, and 2D geometry to procedural surfaces.

Use CONVTOSURFACE to convert any of the following objects into surfaces:

- 2D solids
- Meshes
- Regions
- Open, zero-width polylines with thickness
- Lines with thickness
- Arcs with thickness
- Planar 3D faces

Quick Reference

Commands

CONVTOSURFACE

Converts objects to 3D surfaces.

Create NURBS Surfaces

See also:

- Create Solids and Surfaces from Lines and Curves on page 446
- Create Surfaces from Other Surfaces on page 492
- Edit NURBS Surfaces on page 600
- Rebuild NURBS Surfaces and Curves on page 601
- Draw Splines on page 281
- Modify Splines on page 359

Create NURBS surfaces by enabling NURBS creation and using many of the same commands used to create procedural surfaces. You can also convert existing procedural surfaces into NURBS surfaces.
NURBS (Non-Uniform Rational B-Splines) surfaces are part of the suite of 3D modeling objects that AutoCAD for Mac offers (in addition to 3D solids, procedural surfaces, and meshes).

NURBS surfaces are based on Bezier curves or splines. Therefore, settings such as degree, fit points, control vertices, weights, and knot parameterization are important in defining a NURBS surface or curve. AutoCAD for Mac splines are optimized to create NURBS surfaces allowing you to control many of these options (see SPLINE and SPLINEDIT). The illustration below shows the control vertices that display when you select a NURBS surface or spline.

Two Methods for Creating NURBS Surfaces

There are two ways to create NURBS surfaces:

- **SURFACEMODELINGMODE system variable** - Use any of the surface creation commands while this system variable is set to 1.

- **CONVTONURBS command** - Convert any existing surfaces with this command.

It is important to plan ahead with NURBS modeling, since NURBS surfaces cannot be converted back into procedural surfaces.

Quick Reference

Commands

CONVTONURBS

Converts 3D solids and surfaces into NURBS surfaces.
CVSHOW
Displays the control vertices for specified NURBS surfaces or curves.

EXTRUDE
Creates a 3D solid or surface by extending the dimensions of an object.

LIST
Displays property data for selected objects.

LOFT
Creates a 3D solid or surface in the space between several cross sections.

MEASUREGEOM
Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

REVOLVE
Creates a 3D solid or surface by sweeping an object around an axis.

SPLINE
Creates a smooth B-spline curve that passes through or near a set of points that controls the shape of the curve.

SPLINEDIT
Modifies the parameters of a spline or converts a spline-fit polyline to a spline.

SURFBLEND
Creates a continuous blend surface between two existing surfaces.

SURFEXTEND
Lengthens a surface by a specified distance.

SURFFILLET
Creates a filleted surface between two other surfaces.

SURFNETWORK
Creates a surface in the space between several curves in the U and V directions (including surface and solid edge subobjects).

SURFOFFSET
Creates a parallel surface a specified distance from the original surface.
SURFPATCH
Creates a new surface by fitting a cap over a surface edge that forms a closed loop.

SWEEP
Creates a 3D solid or surface by sweeping a 2D or 3D object or subobject along a path.

System Variables
SURFACEMODELINGMODE
Controls whether surfaces are created as procedural surfaces or NURBS surfaces.

Using the Spline tool to create NURBS Surfaces and Splines

See also:
- Draw Splines on page 281
- Modify Splines on page 359

The spline tool is optimized to work with NURBS modeling
NURBS surfaces can be created from a number of 2D objects, including edge subobjects, polylines, and arcs. But the spline tool is the only object that has options that are compatible with creating NURBS surface. Not only do splines consist of Bezier arcs, but they also can be defined with both control vertices and fit points. The fit points and control vertices offer different editing options such as knot parameterization and degree options.

Moving Fit Points vs. Moving Control Vertices
NURBS curves have both fit points and control vertices. The fit points lie on the line, and the control vertices lay outside the line. Use fit points to make a change to one small part of a curve; use control vertices to make changes that will affect the shape of the curve as a whole.
Clamp Surfaces and Curves with Open and Closed Geometry

NURBS surfaces and curves can have a clamp, closed, or open form. The form affects how the object deforms.

- **Open Curves and surfaces** - have their start and end CVs in different positions - it doesn’t form a loop. If you snap the start and end CVs of an open curve to the same position, it’s still an open curve, because you can still drag these points away from one another.

- **Closed Curves and Surfaces** - a loop with coinciding start and end CVs. Where they meet is called a seam. If you move one CV, the other moves with it.

- **Clamp Curve** - is a closed loop with a seam that creates extra, unseen CVs. These unseen CVs can cause the shape to wrinkle and crease when it is reshaped.

Quick Reference

**Commands**

SPLINE

Creates a smooth B-spline curve that passes through or near a set of points that controls the shape of the curve.

SPLINEDIT

Modifies the parameters of a spline or converts a spline-fit polyline to a spline.

**System Variables**

SURFACEMODELINGMODE

Controls whether surfaces are created as procedural surfaces or NURBS surfaces.

Create Associative Surfaces

Associative surfaces automatically adjust to changes made to other, related objects.

When surface associativity is on, surfaces are created with a relationship to the surface or profiles that created them.
Associativity allows you to:

- Reshape the generating profiles to automatically reshape the surface.
- Work with a group of surfaces as if they were one object. Just as reshaping one face of a solid box adjusts the entire primitive, reshaping one surface or edge in a group of associated surfaces adjusts the entire group.
- Use geometric constraints on the 2D profiles of a surface.
- Assign mathematical expressions to derive properties of surfaces, such as height and radius. For example, specify that the height of an extruded surface be equal to one half the length of another object.

As you add more objects and edit them, all these objects become related and create a chain of dependency. Editing one object can ripple through and affect all associated objects.

It is important to understand the chain of associativity because moving or deleting one of the links in the chain can break the relationship between all the objects.

**NOTE** To modify the shape of a surface that is generated from a curve or spline, you must select and modify the generating curve or spline, not the surface itself. If you modify the surface itself, you will lose associativity.

When associativity is on, the DELOBJ system variable is ignored. If Surface Associativity and NURBS Creation are both on, surfaces are created as NURBS surfaces, not associative surfaces.

Save time by planning your model ahead; you cannot go back and add associativity after the model has been created. Also, be careful not to accidentally break associativity by dragging objects away from the group.

**Quick Reference**

**Commands**

BREP

Removes the history from 3D solids and composite solids, and associativity from surfaces.

EXTRUDE

Creates a 3D solid or surface by extending the dimensions of an object.
JOIN

Joins the endpoints of lines, 2D and 3D polylines, arcs, elliptical arcs, helixes, and splines to create single object.

LIST

Displays property data for selected objects.

LOFT

Creates a 3D solid or surface in the space between several cross sections.

MEASUREGEOM

Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

PLANESURF

Creates a planar surface.

PROJECTGEOMETRY

Projects points, lines, or curves onto a 3D solid or surface from different directions.

REVOLVE

Creates a 3D solid or surface by sweeping an object around an axis.

SURFBLND

Creates a continuous blend surface between two existing surfaces.

SURFNETWORK

Creates a surface in the space between several curves in the U and V directions (including surface and solid edge subobjects).

SURFOFFSET

Creates a parallel surface a specified distance from the original surface.

SURFPATCH

Creates a new surface by fitting a cap over a surface edge that forms a closed loop.

SWEEP

Creates a 3D solid or surface by sweeping a 2D or 3D object or subobject along a path.
VISUALSTYLES
  Creates and modifies visual styles and applies a visual style to a viewport.

System Variables
SURFACEASSOCIATIVITY
  Controls whether surfaces maintain a relationship with the objects from which they were created.
SURFACEASSOCIATIVITYDRAG
  Sets the dragging preview behavior of associative surfaces.

Create Geometric Relationships between Associative Surfaces
  Use geometric constraints to constrain and restrict surfaces. And use mathematical expressions to derive surface properties.

Use Geometric Constraints with Surface Profiles
  Just as with 2D drafting, geometric constraints can be used to restrict the movement of 3D surfaces. For example, you can specify that a surface remain fixed in a perpendicular or parallel location to another object. In the example below, an offset surface is locked in a parallel position to its original surface.

Constraints are applied to the 2D profile object used to create the surface, not the surface itself. Use selection cycling to be sure that you are selecting the profile curve and not the surface or the edge subobject. See Apply or Remove Geometric Constraints on page 373.

Use Mathematical Expressions to Derive Surface Properties
  Dimensional constraints are user-defined expressions that are applied in the Properties Inspector for that surface.
For a complete list of operators and functions allowed in expressions, see Control Geometry with Parameters on page 399. The following table lists the surface types and their properties that accept expressions:

<table>
<thead>
<tr>
<th>Type of Surface</th>
<th>Surface properties that can be constrained.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Blend Surface</td>
<td>Bulge Magnitude</td>
</tr>
<tr>
<td>Extend Surface</td>
<td>Extension Distance</td>
</tr>
<tr>
<td>Extruded Surface</td>
<td>■ Height</td>
</tr>
<tr>
<td></td>
<td>■ Taper</td>
</tr>
<tr>
<td>Fillet Surface</td>
<td>Fillet Radius</td>
</tr>
<tr>
<td>Loft Surface</td>
<td>Bulge Magnitude</td>
</tr>
<tr>
<td>Network Surface</td>
<td>Bulge Magnitude</td>
</tr>
<tr>
<td>Offset Surface</td>
<td>Offset Distance</td>
</tr>
<tr>
<td>Patch Surface</td>
<td>Bulge Magnitude</td>
</tr>
<tr>
<td>Revolve Surface</td>
<td>Revolve Angle</td>
</tr>
</tbody>
</table>

**Quick Reference**

**Commands**

**AUTOCONSTRAIN**

Applies geometric constraints to a selection set of objects based on orientation of the objects relative to one another.

**CONSTRAINTBAR**

Displays or hides the geometric constraints on an object.

**CONSTRAINTSETTINGS**

Controls the display of geometric constraints on constraint bars.
DCDISPLAY
Displays or hides the dynamic constraints associated with a selection set of objects.

DELCONSTRAINT
Removes all geometric and dimensional constraints from a selection set of objects.

DIMCONSTRAINT
Applies dimensional constraints to selected objects or points on objects, or converts associative dimensions to dimensional constraints.

GEOMCONSTRAINT
Displays or hides the geometric constraints on an object.

LIST
Displays property data for selected objects.

MEASUREGEOM
Measures the distance, radius, angle, area, and volume of selected objects or sequence of points.

PARAMETERS (-PARAMETERS)
Controls the associative parameters used in the drawing.

TEXTEDIT
Edits a dimensional constraint, dimension, or text object.

**System Variables**

**CCONSTRAINTFORM**
Controls whether annotational or dynamic constraints are applied to objects.

**CONSTRAINTBARDISPLAY**
Controls the display of constraint bars after you apply constraints and when you select geometrically constrained drawings.

**CONSTRAINTBARMODE**
Controls the display of geometrical constraints on constraint bars.

**CONSTRAINTNAMEFORMAT**
Controls the text format for dimensional constraints.
CONSTRANTRELAX
Indicates whether constraints are enforced or relaxed when editing an object.

CONSTRAINTSOLVEMODE
Controls constraint behavior when applying or editing constraints.

DIMCONSTRAINTICON
Displays the lock icon next to the text for dimensional constraints.

DYNCONSTRAINTMODE
Displays hidden dimensional constraints when constrained objects are selected.

PARAMETERCOPYMODE
Controls how constraints and referenced user parameters are handled when constrained objects are copied between drawings, Model space and layouts, and block definitions.

SURFACEASSOCIATIVITY
Controls whether surfaces maintain a relationship with the objects from which they were created.

Create Meshes
Create meshes from primitive forms or by filling between points on other objects.

Overview of Creating Meshes
Mesh tessellation provides enhanced capabilities for modeling object shapes in a more detailed way.
Starting with AutoCAD 2010, the default mesh object type can be smoothed, creased, split, and refined. Although you can continue to create the legacy polyface and polygon mesh types, you can obtain more predictable results by converting to the newer mesh object type.

**Methods for Creating Mesh**

You can create mesh objects using the following methods:

- **Create mesh primitives.** Create standard shapes, such as boxes, cones, cylinders, pyramids, spheres, wedges, and tori (MESH).

- **Create mesh from other objects.** Create ruled, tabulated, revolved, or edge-defined mesh objects, whose boundaries are interpolated from other objects or points (RULESURF, TABSURF, REVSURF, EDGESURF).

- **Convert from other object types.** Convert existing solid or surface models, including composite models, to mesh objects (MESHSMOOTH). You can also convert the legacy style of mesh to the new mesh object type.

- **Create custom meshes (legacy).** Use 3DMESH to create polygon meshes, usually scripted with AutoLISP routines, to create open-ended mesh. Use PFACE to create mesh with multiple vertices defined by coordinates that you specify. Although you can continue to create legacy polygonal and polyface meshes, it is recommended that you convert to the enhanced mesh object type to obtain enhanced editing capabilities.

**About Tessellation**

Tessellation is a collection of planar shapes that tile a mesh object. The tessellation divisions, visible in unselected mesh objects, mark the edges of the editable mesh faces. (To see these divisions in the Hidden or Conceptual visual styles, VSEDGES must be set to 1.)
When you smooth and refine mesh objects, you increase the density of the tessellation (the number of subdivisions).

- **Smoothing.** Increases how closely the mesh surface adheres to a rounded form. You can increase mesh smoothness levels for selected objects in increments or by changing the smoothness level in the Properties Inspector. Smoothness level 0 (zero) applies the lowest level of smoothing to a mesh object. Smoothness level 4 applies a high degree of smoothness.

- **Refinement.** Quadruples the number of subdivisions in a selected mesh object or in a selected subobject, such as a face. Refinement also resets the current smoothness level to 0, so that the object can no longer be sharpened beyond that level. Because refinement greatly increases the density of a mesh, you might want to restrict this option to areas that require finely detailed modification. Refinement also helps you mold smaller sections with less effect on the overall shape of the model.
While highly refined mesh gives you the ability to make detailed modifications, it also comes at a cost: it can decrease program performance. By maintaining maximum smoothness, face, and grid levels, you can help ensure that you do not create meshes that are too dense to modify effectively. (Use SMOOTHMESHMAXLEV, SMOOTHMESHMAXFACE, and SMOOTHMESHGRID.)

**Set Mesh Properties Before and After Creation**

You can set defaults that control a variety of mesh properties before and after you create the mesh objects.

- **Properties Inspector.** Modifies properties for both the mesh object and its subobjects after they are created. For a selected mesh object, you can modify the level of smoothness. For faces and edges, you can apply or remove creasing, and modify crease retention levels.

- **Level of smoothness.** By default, the mesh primitive objects that you create have no smoothness. You can change this default with the Settings option of the MESH command. The modified smoothness value is maintained only during the current drawing session.

See also:

- [Modify Mesh Objects on page 610](#)

**Quick Reference**

**Commands**

**3DFACE**

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

**3DMESH**

Creates a free-form polygon mesh.

**EDGESURF**

Creates a mesh between four contiguous edges or curves.

**MESH**

Creates a 3D mesh primitive object such as a box, cone, cylinder, pyramid, sphere, wedge, or torus.
MESHSMOOTH
Converts 3D objects such as polygon meshes, surfaces, and solids to mesh objects.

PFACE
Creates a 3D polyface mesh vertex by vertex.

PROPERTIES
Controls properties of existing objects.

REVSURF
Creates a mesh by revolving a profile about an axis.

RULESURF
Creates a mesh that represents the surface between two lines or curves.

TABSURF
Creates a mesh from a line or curve that is swept along a straight path.

System Variables
SMOOTHMESHMAXLEV
Sets the maximum smoothness level for mesh objects.

SMOOTHMESHMAXFACE
Sets the maximum number of faces permitted for mesh objects.

SMOOTHMESHGRID
Sets the maximum level of smoothness at which the underlying mesh facet grid is displayed on 3D mesh objects.

VSEDGES
Controls the types of edges that are displayed in the viewport.

Create 3D Mesh Primitives
Create mesh boxes, cones, cylinders, pyramids, spheres, wedges, and tori.

Create a Mesh Box
Create a rectangular or cubical mesh box.
The base of the mesh box is drawn parallel to the $XY$ plane of the current UCS (workplane).

The following system variables are used to control the number of divisions for each dimension of a new mesh box:

- DIVMESHBOXHEIGHT
- DIVMESHBOXLENGTH
- DIVMESHBOXWIDTH

After a mesh primitive is created, the current level of smoothness for the object can be modified.

**Mesh Box Creation Options**

The Box option of the MESH command provides several methods for determining the size and rotation of the mesh boxes you create.

- **Create a cube.** Use the Cube option to create a mesh box with sides of equal length.
- **Specify rotation.** Use the Cube or Length option if you want to set the rotation of the box in the $XY$ plane.
- **Start from the center point.** Use the Center option to create a box using a specified center point.

**Quick Reference**

**Commands**

MESH

Creates a 3D mesh primitive object such as a box, cone, cylinder, pyramid, sphere, wedge, or torus.
**System Variables**

**DIVMESHBOXHEIGHT**
Sets the number of subdivisions for the height of a mesh box along the Z axis.

**DIVMESHBOXLENGTH**
Sets the number of subdivisions for the length of a mesh box along the X axis.

**DIVMESHBOXWIDTH**
Sets the number of subdivisions for the width of a mesh box along the Y axis.

**DRAGVS**
Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

**Create a Mesh Cone**

Create a pointed or frustum mesh cone with a circular or elliptical base.

By default, the base of the mesh cone lies on the XY plane of the current UCS and the height of the cone is parallel to the Z axis.

The following system variables are used to control the number of divisions for each dimension of a new mesh cone:

- **DIVMESHCONEAAXIS**
- **DIVMESHCONEBASE**
- **DIVMESHCONEHIGHT**

After a mesh primitive is created, the current level of smoothness for the object can be modified.
Mesh Cone Creation Options

The Cone option of the MESH command provides several methods for determining the size and rotation of the mesh cones you create.

- **Set the height and orientation.** Use the Axis Endpoint option when you want to reorient the cone by placing the tip or axis endpoint anywhere in 3D space.

- **Create a frustum of a cone.** Use the Top Radius option to create a frustum of a cone, which tapers to an elliptical or planar face.

- **Specify circumference and base plane.** The 3P (Three Points) option defines the size and plane of the base of the cone anywhere in 3D space.

- **Create an elliptical base.** Use the Elliptical option to create a cone base whose axes are different lengths.

- **Set the location to be tangent to two objects.** Use the Ttr (Tangent, Tangent, Radius) option to define points on two objects. Depending on the radius distance, the new cone is located as near as possible to the tangent points you specify. You can set up tangency with circles, arcs, lines, and some 3D objects. The tangency points are projected onto the current UCS. The appearance of tangency is affected by the current level of smoothness.

Quick Reference

**Commands**

MESH

Creates a 3D mesh primitive object such as a box, cone, cylinder, pyramid, sphere, wedge, or torus.

**System Variables**

DIVMESHCONEAXIS

Sets the number of subdivisions around the perimeter of the mesh cone base.

DIVMESHCONEBASE

Sets the number of subdivisions between the perimeter and the center point of the mesh cone base.
DIVMESHCONENHEIGHT
Sets the number of subdivisions between the base and the point or top of the mesh cone.

DRAGVS
Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Create a Mesh Cylinder
Create a mesh cylinder with a circular or elliptical base.

By default, the base of the mesh cylinder lies on the $XY$ plane of the current UCS. The height of the cylinder is parallel to the $Z$ axis.

The following system variables are used to control the number of divisions for each dimension of a new mesh cylinder:

- DIVMESHCYLAXIS
- DIVMESHCYLBASE
- DIVMESHCYLHEIGHT

After a mesh primitive is created, the current level of smoothness for the object can be modified.
**Mesh Cylinder Creation Options**

The Cylinder option of the MESH command provides several methods for determining the size and rotation of the mesh cylinders you create.

- **Set rotation.** Use the Axis Endpoint option to set the height and rotation of the cylinder. The center point of the top plane of the cylinder is the axis endpoint, which can be located anywhere in 3D space.

- **Use three points to define the base.** Use the 3P (Three Points) option to define the base of the cylinder. You can set three points anywhere in 3D space.

- **Create an elliptical base.** Use the Elliptical option to create a cylinder base whose axes are different lengths.

- **Set the location to be tangent to two objects.** Use the Ttr (Tangent, Tangent, Radius) option to define points on two objects. Depending on the radius distance, the new cylinder is located as near as possible to the tangent points you specify. You can set up tangency with circles, arcs, lines, and some 3D objects. The tangency points are projected onto the current UCS. The appearance of tangency is affected by the current level of smoothness.

**Quick Reference**

**Commands**

MESH

Creates a 3D mesh primitive object such as a box, cone, cylinder, pyramid, sphere, wedge, or torus.

**System Variables**

DIVMESHCYLAXIS

Sets the number of subdivisions around the perimeter of the mesh cylinder base.

DIVMESHCYLBASE

Sets the number of radial subdivisions from the center of the mesh cylinder base to its perimeter.
DIVMESHCYLHEIGHT
Sets the number of subdivisions between the base and the top of the mesh cylinder.

DRAGVS
Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Create a Mesh Pyramid
Create a mesh pyramid with up to 32 sides.

Create a pyramid that tapers to a point, or create a frustum of a pyramid, which tapers to a planar face.

The following system variables are used to control the number of divisions for each dimension of a new mesh pyramid:

- DIVMESHPYRBASE
- DIVMESHPYRHEIGHT
- DIVMESHPYRLENGTH

After a mesh primitive is created, the current level of smoothness for the object can be modified.

Mesh Pyramid Creation Options
The Pyramid option of the MESH command provides several methods for determining the size and rotation of the mesh pyramids you create.

- **Set the number of sides.** Use the Sides option to set the number of sides for the mesh pyramid.

- **Set the length of the edges.** Use the Edges option to specify the dimension of the sides at the base.
■ **Create a frustum of a pyramid.** Use the Top Radius option to create a frustum, which tapers to a planar face. The frustum face is parallel to, and has the same number of sides as, the base.

![Frustum of a pyramid](image)

■ **Set the height and rotation of the pyramid.** Use the Axis Endpoint option to specify the height and rotation of the pyramid. This endpoint is the top of the pyramid. The axis endpoint can be located anywhere in 3D space.

■ **Set the perimeter to be inscribed or circumscribed.** Specify whether the pyramid base is drawn inside or outside of the radius.

![Inscribed and circumscribed pyramids](image)

**Quick Reference**

**Commands**

MESH

Creates a 3D mesh primitive object such as a box, cone, cylinder, pyramid, sphere, wedge, or torus.
**System Variables**

**DIVMESHPYRBASE**
Sets the number of radial subdivisions between the center of the mesh pyramid base and its perimeter.

**DIVMESHPYRHEIGHT**
Sets the number of subdivisions between the base and the top of the mesh pyramid.

**DIVMESHPYRENGTH**
Sets the number of subdivisions along each dimension of a mesh pyramid base.

**DRAGVS**
Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

---

**Create a Mesh Sphere**

Create a mesh sphere using one of several methods.

When you start with the center point, the central axis of the mesh sphere parallels the Z axis of the current user coordinate system (UCS).

The following system variables are used to control the number of divisions for each dimension of a new mesh sphere:

- **DIVMESHSPHEREAXIS**
- **DIVMESHSPHEREHEIGHT**

After a mesh primitive is created, the current level of smoothness for the object can be modified.
Mesh Sphere Creation Options

The Sphere option of the MESH command provides several methods for determining the size and rotation of the mesh spheres you create.

■ Specify three points to set the size and plane of the circumference or radius. Use the 3P (Three Points) option to define the size of the sphere anywhere in 3D space. The three points also define the plane of the circumference.

■ Specify two points to set the circumference or radius. Use the 2P (Two Points) option to define the size of the sphere anywhere in 3D space. The plane of the circumference matches the Z value of the first point.

■ Set the location to be tangent to two objects. Use the Ttr (Tangent, Tangent, Radius) option to define points on two objects. Depending on the radius distance, the sphere is located as near as possible to the tangent points you specify. You can set up tangency with circles, arcs, lines, and some 3D objects. The tangency points are projected onto the current UCS. The appearance of tangency is affected by the current level of smoothness.

Quick Reference

Commands
MESH

Creates a 3D mesh primitive object such as a box, cone, cylinder, pyramid, sphere, wedge, or torus.

System Variables
DIVMESHSPHEREAXIS

Sets the number of radial subdivisions around the axis endpoint of the mesh sphere.

DIVMESHSPHEREHEIGHT

Sets the number of subdivisions between the two axis endpoints of the mesh sphere.

DRAGVS

Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.
Create a Mesh Wedge

Create a mesh wedge with rectangular or cubical faces.

The base of the wedge is drawn parallel to the XY plane of the current UCS with the sloped face opposite the first corner. The height of the wedge is parallel to the Z axis.

The following system variables are used to control the number of divisions for each dimension of a new mesh wedge:

- `DIVMESHWEDGEBASE`
- `DIVMESHWEDGEHEIGHT`
- `DIVMESHWEDGELENGTH`
- `DIVMESHWEDGESLOPE`
- `DIVMESHWEDGEWIDTH`

After a mesh primitive is created, the current level of smoothness for the object can be modified.

Mesh Wedge Creation Options

The Wedge option of the `MESH` command provides several methods for determining the size and rotation of the mesh wedges you create.

- **Create a wedge with sides of equal length.** Use the Cube option.
- **Specify rotation.** Use the Cube or Length option if you want to set the rotation of the mesh wedge in the XY plane.
- **Start from the center point.** Use the Center Point option.
Quick Reference

Commands

MESH

Creates a 3D mesh primitive object such as a box, cone, cylinder, pyramid, sphere, wedge, or torus.

System Variables

DIVMESHWEDGEBASE

Sets the number of subdivisions between the midpoint of the perimeter of triangular dimension of the mesh wedge.

DIVMESHWEDGEHEIGHT

Sets the number of subdivisions for the height of the mesh wedge along the Z axis.

DIVMESHWEDGELENGTH

Sets the number of subdivisions for the length of a mesh wedge along the X axis.

DIVMESHWEDGESLOPE

Sets the number of subdivisions in the slope that extends from the apex of the wedge to the edge of the base.

DIVMESHWEDGEWIDTH

Sets the number of subdivisions for the width of the mesh wedge along the Y axis.

DRAGVS

Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Create a Mesh Torus

Create a ring-shaped solid that resembles the inner tube of a tire.
A mesh torus has two radius values. One value defines the tube. The other value defines the path, which is equivalent to the distance from the center of the torus to the center of the tube. By default, a torus is drawn parallel to and is bisected by the $XY$ plane of the current UCS.

A mesh torus can be self-intersecting. A self-intersecting mesh torus has no center hole because the radius of the tube is greater than the radius of the torus.

The following system variables are used to control the number of divisions for each dimension of a new mesh torus:

- `DIVMESHTORUSPATH`
- `DIVMESHTORUSSECTION`

After a mesh primitive is created, the current level of smoothness for the object can be modified.

**Torus Creation Options**

The Torus option of the MESH command provides several methods for determining the size and rotation of the mesh tori you create.

- **Set the size and plane of the circumference or radius.** Use the 3P (Three Points) option to define the size of the mesh torus anywhere in 3D space. The three points also define the plane of the circumference. Use this option to rotate the mesh torus as you create it.

- **Set the circumference or radius.** Use the 2P (Two Points) option to define the size of the mesh torus anywhere in 3D space. The plane of the circumference matches the $Z$ value of the first point.

- **Set the location to be tangent to two objects.** Use the Ttr (Tangent, Tangent, Radius) option to define points on two objects. Depending on the specified radius distance, the path of the torus is located as near as possible to the tangent points you specify. You can set up tangency with circles, arcs, lines, and some 3D objects. The tangency points are projected
onto the current UCS. The appearance of tangency is affected by the current level of smoothness.

Quick Reference

Commands
MESH
Creates a 3D mesh primitive object such as a box, cone, cylinder, pyramid, sphere, wedge, or torus.

System Variables
DIVMESHTORUSPATH
Sets the number of subdivisions in the path that is swept by the profile of a mesh torus.
DIVMESHTORUSSECTION
Sets the number of subdivisions in the profile that sweeps the path of a mesh torus.
DRAGVS
Sets the visual style that is displayed while creating 3D solid and mesh primitives and extruded solids, surfaces, and meshes.

Construct Meshes from Other Objects
Create mesh forms by filling the space between other objects such as lines and arcs.

You can use a variety of methods to create mesh objects whose edges are defined by other objects. The MESHTYPE system variable controls whether the new objects are valid mesh objects, or whether they are created using legacy polyface or polygon geometry.

You can control whether the mesh is displayed as a wireframe, hidden, or conceptual image by changing the visual style (-VISUALSTYLES).
Types of Meshes Created from Other Objects

You can create several types of meshes that are based on existing objects.

- **Ruled mesh.** RULESURF creates a mesh that represents the ruled surface between two lines or curves.

- **Tabulated mesh.** TABSURF creates a mesh that represents a general tabulated surface. The surface is defined by the extrusion of a line or curve (called a path curve) in a specified direction and distance (called a direction vector or path).

- **Revolved mesh.** REVSURF creates a mesh that approximates a surface of revolution by rotating a profile about a specified axis. A profile can consist of lines, circles, arcs, ellipses, elliptical arcs, polylines, splines, closed polylines, polygons, closed splines, and donuts.
**Edge-defined mesh.** EDGESURF creates a mesh approximating a Coons surface patch mesh from four adjoining edges. A Coons surface patch mesh is a bicubic surface that is interpolated between four adjoining edges (which can be general space curves).

**Create a Ruled Mesh**

There are several methods for creating meshes.

With RULESURF, you create a mesh between two lines or curves. Use two different objects to define the edges of the ruled mesh: lines, points, arcs, circles, ellipses, elliptical arcs, 2D polylines, 3D polylines, or splines.

Both objects that are used as the “rails” of a ruled mesh must be either open or closed. You can pair a point object with either an open or a closed object.
You can specify any two points on closed curves to complete the operation. For open curves, construction of the ruled mesh is based on the locations of the specified points on the curves.

Create a Tabulated Mesh

With the TABSURF command, you can create a mesh that represents a general tabulated surface defined by a path curve and a direction vector. The path curve can be a line, arc, circle, ellipse, elliptical arc, 2D polyline, 3D polyline, or spline. The direction vector can be a line or an open 2D or 3D polyline.

TABSURF creates the mesh as a series of parallel polygons running along a specified path. The original object and the direction vector must already be drawn, as shown in the following illustrations.

Create a Revolved Mesh

Use the REVSURF command to create a revolved mesh by rotating a profile of the object about an axis. REVSURF is useful for mesh forms with rotational symmetry.
The profile is called a path curve. It can consist of any combination of lines, circles, arcs, ellipses, elliptical arcs, polylines, splines, closed polylines, polygons, closed splines, or donuts.

**Create an Edge-Defined Mesh**

With the EDGESURF command, you can create a Coons surface patch mesh, as shown in the following illustration, from four objects called edges. Edges can be arcs, lines, polylines, splines, or elliptical arcs that form a closed loop and share endpoints. A Coons patch is a bicubic surface (one curve in the M direction and another in the N direction) interpolated between the four edges.

**Quick Reference**

**Commands**

**EDGESURF**

Creates a mesh between four contiguous edges or curves.

**PEDIT**

Edits polylines and 3D polygon meshes.

**PFACE**

Creates a 3D polyface mesh vertex by vertex.

**REVSURF**

Creates a mesh by revolving a profile about an axis.
RULESURF
 Creates a mesh that represents the surface between two lines or curves.

TABSURF
 Creates a mesh from a line or curve that is swept along a straight path.

VISUALSTYLES (-VISUALSTYLES)
 Creates and modifies visual styles and applies a visual style to a viewport.

**System Variables**

FACETRATIO
 Controls the aspect ratio of faceting for cylindrical and conic solids.

MESHTYPE
 Controls the type of mesh that is created by REVSURF, TABSURF, RULESURF and EDGESURF.

PLINECONVERTMODE
 Specifies the fit method used in converting splines to polylines.

PFACEVMAX
 Sets the maximum number of vertices per face.

SURFTAB1
 Sets the number of tabulations to be generated for the RULESURF and TABSURF commands.

SURFTAB2
 Sets the mesh density in the N direction for the REVSURF and EDGESURF commands.

SURFTYPE
 Controls the type of surface-fitting to be performed by the Smooth option of the PEDIT command.

SURFU
 Sets the surface density for PEDIT Smooth in the M direction and the U isolines density on surface objects.
SURFV

Sets the surface density for PEDIT Smooth in the N direction and the V isolines density on surface objects.

Create Meshes by Conversion

Convert solids, surfaces, and legacy mesh types to mesh objects.

You can use the MESHSMOOTH command to convert certain objects to mesh. Convert 3D solids, surfaces, and legacy mesh objects to the enhanced mesh object in order to take advantage of capabilities such as smoothing, refinement, creasing, and splitting.

Object Types That Can Be Converted

You obtain the most predictable results when you convert primitive solid objects to mesh. That is, the resulting mesh adheres closely to the shape of the original solid model.

You can also convert other types of objects, although the conversion results may differ from what you expect. These objects include swept surfaces and solids, legacy polygon and polyface mesh objects, regions, closed polylines, and objects created with 3DFACE. For these objects, you can often improve results by adjusting the conversion settings.

Adjust Mesh Conversion Settings

If the conversion does not work as expected, try changing the following system variables:

- FACETERDEVNORM
- FACETERDEVSURFACE

534 | Chapter 22  Create 3D Models
For example, if the smooth mesh optimized mesh type (FACETERMESHTYPE system variable) results in incorrect conversions, you can set the tessellation shape to be Triangle or Mostly Quads.

You also can control the adherence to the original shape by setting the maximum distance offset, angles, aspect ratios, and edge lengths for new faces. The following example shows a 3D solid helix that has been converted to mesh using different tessellation settings. The optimized mesh version has been smoothed, but the other two conversions have no smoothness. Notice, however, that the mostly quads conversion with the lower tessellation values creates a mesh object that adheres most closely to the original version. Smoothing this object improves its appearance even more.

Similarly, if you notice that a converted mesh object has a number of long, slivered faces (which can sometimes cause gaps), try decreasing the maximum edge length for new faces value (FACETERMAXEDGELENGTH system variable).

If you are converting primitive solid objects, this dialog box also offers the option of using the same default settings used to create primitive mesh objects.
When you select conversion candidates directly from this dialog box, you can preview the results before you accept them.

See also:
- Objects That Can Be Converted to Mesh

Quick Reference

Commands
MESHOPTIONS
Displays the Mesh Tessellation Options dialog box, which controls default settings for converting existing objects to mesh objects.

MESHSMOOTH
Converts 3D objects such as polygon meshes, surfaces, and solids to mesh objects.

System Variables
FACETERDEVNORMAL
Sets the maximum angle between the surface normal and contiguous mesh faces.

FACETERDEVSURFACE
Sets how closely the converted mesh object adheres to the original shape of the solid or surface.

FACETERGRIDRATIO
Sets the maximum aspect ratio for the mesh subdivisions that are created for solids and surfaces converted to mesh.

FACETERMAXEDGELENGTH
Sets the maximum length of edges for mesh objects that are created by conversion from solids and surfaces.

FACETERMAXGRID
Sets the maximum number of U and V grid lines for solids and surfaces converted to mesh.
FACETERMESHTYPE
Sets the type of mesh to be created.

FACETERMINUGRID
Sets the minimum number of U grid lines for solids and surfaces that are converted to mesh.

FACETERMINVGRID
Sets the minimum number of V grid lines for solids and surfaces that are converted to mesh.

FACETERSMOOTHLEV
Sets the default level of smoothness for objects that are converted to mesh.

Create Custom Mesh (Legacy)
Create custom polygon or polyface mesh by specifying vertices.
Specify individual vertices when you create mesh using the 3DMESH, PFACE, and 3DFACE commands.

Understand Legacy Mesh Construction
The mesh density controls the number of facets in legacy polygonal and polyface meshes. Density is defined in terms of a matrix of $M$ and $N$ vertices, like a grid consisting of columns and rows. $M$ and $N$ specify the column and row position, respectively, of any given vertex.
A mesh can be open or closed. If the start and end edges of the mesh do not touch, a mesh is open in a given direction, as shown in the following illustrations.
Create a Rectangular Mesh

With the 3DMESH command, you can create polygon meshes that are open in both the \( M \) and \( N \) directions (like the \( X \) and \( Y \) axes of an \( XY \) plane). In most cases, you can use 3DMESH in conjunction with scripts or AutoLISP routines when you know the mesh points.

As you create the mesh, you specify the size of the mesh in the \( M \) and \( N \) directions. The total number of vertices you specify for the mesh is equal to the \( M \) value times the \( N \) value.
You can close the meshes with PEDIT. You can use 3DMESH to construct irregular meshes.

**Example:**

In the following example of text at the Command prompt, you enter the coordinate values for each vertex to create the mesh in the illustration.

**Command:** `3dmesh`
**Mesh M size:** 4
**Mesh N size:** 3
**Vertex (0, 0):** 10, 1, 3
**Vertex (0, 1):** 10, 5, 5
**Vertex (0, 2):** 10, 10, 3
**Vertex (1, 0):** 15, 1, 0
**Vertex (1, 1):** 15, 5, 0
**Vertex (1, 2):** 15, 10, 0
**Vertex (2, 0):** 20, 1, 0
**Vertex (2, 1):** 20, 5, -1
**Vertex (2, 2):** 20, 10, 0
**Vertex (3, 0):** 25, 1, 0
**Vertex (3, 1):** 25, 5, 0
**Vertex (3, 2):** 25, 10, 0

![Mesh diagram](image)

**Create a Polyface Mesh**

The PFACE command produces a polyface (polygon) mesh, with each face capable of having numerous vertices. PFACE is typically used by applications rather than by direct user input.

Creating a polyface mesh is like creating a rectangular mesh. To create a polyface mesh, you specify coordinates for its vertices. You then define each face by entering vertex numbers for all the vertices of that face. As you create
the polyface mesh, you can set specific edges to be invisible, assign them to layers, or give them colors.

To make the edge invisible, enter the vertex number as a negative value. For instance, to make the edge between vertices 5 and 7 invisible in the following illustration, you enter the following:

Face 3, vertex 3: -7

In the illustration, face 1 is defined by vertices 1, 5, 6, and 2. Face 2 is defined by vertices 1, 4, 3, and 2. Face 3 is defined by vertices 1, 4, 7, and 5, and face 4 is defined by vertices 3, 4, 7, and 8.

Create Polyface Mesh Vertex by Vertex

With the 3DFACE command, you can create three-dimensional polyface mesh by specifying each vertex. You can control visibility of each mesh edge segment.

If you select a 3DFACE object during some mesh smoothing operations (such as with MESHSMOOTHMORE), you are prompted to convert 3DFACE objects to mesh objects.

Create a Predefined 3D Mesh

The 3D command creates the following 3D shapes: boxes, cones, dishes, domes, meshes, pyramids, spheres, tori (donuts), and wedges.

In the following illustrations, the numbers indicate points you specify to create the mesh.
To view the objects you are creating with the 3D command more clearly, set a viewing direction with 3DORBIT, DVIEW, or VPOINT.

**Quick Reference**

**Commands**

3DFACE

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

3DMESH

Creates a free-form polygon mesh.

PEDIT

Edits polylines and 3D polygon meshes.

PFACE

Creates a 3D polyface mesh vertex by vertex.
System Variables

PFACEVMAX

Sets the maximum number of vertices per face.

Create Wireframe Models

A wireframe model is an edge or skeletal representation of a real-world 3D object using lines and curves.

You can specify a wireframe visual style to help you see the overall structure of 3D objects such as solids, surfaces, and meshes. In older drawings, you might also encounter wireframe models that were created using legacy methods.

Wireframe models consist only of points, lines, and curves that describe the edges of the object. Because each object that makes up a wireframe model must be independently drawn and positioned, this type of modeling can be the most time-consuming.

You can use a wireframe model to

- View the model from any vantage point
- Generate standard orthographic and auxiliary views automatically
- Generate exploded and perspective views easily
Analyze spatial relationships, including the shortest distance between corners and edges, and checking for interferences

Reduce the number of prototypes required

The ISOLINES system variable controls the number of tessellation lines used to visualize curved portions of the wireframe. The FACETRES system variable adjusts the smoothness of shaded and hidden-line objects.

**Methods for Creating Wireframe Models**

You can create wireframe models by positioning any 2D planar object anywhere in 3D space, using the following methods:

- Use the XEDGES command to create wireframe geometry from regions, 3D solids, surfaces, and meshes. XEDGES extracts all the edges on the selected objects or subobjects. The extracted edges form a duplicate wireframe composed of 2D objects such as lines, circles, and 3D polylines.
- Enter 3D coordinates that define the $X$, $Y$, and $Z$ location of the object.
- Set the default workplane (the XY plane of the UCS) on which to draw the object.
- Move or copy the object to its proper 3D location after you create it.

Wireframe modeling is a skill that requires practice and experience. The best way to learn how to create wireframe models is to begin with simple models before attempting models that are more complex.

**Tips for Working with Wireframe Models**

Creating 3D wireframe models can be more difficult and time-consuming than creating their 2D views. Here are some tips that will help you work more effectively:

- Plan and organize your model so that you can turn off layers to reduce the visual complexity of the model. Color can help you differentiate between objects in various views.
- Create construction geometry to define the basic envelope of the model.
- Use multiple views, especially isometric views, to make visualizing the model and selecting objects easier.
- Become adept at manipulating the UCS in 3D. The XY plane of the current UCS operates as a workplane to orient planar objects such as circles and
arcs. The UCS also determines the plane of operation for trimming and extending, offsetting, and rotating objects.

- Use object snaps and grid snap carefully to ensure the precision of your model.
- Use coordinate filters to drop perpendiculars and easily locate points in 3D based on the location of points on other objects.

**Quick Reference**

**Commands**

**UCS**
Manages user coordinate systems.

**XEDGES**
Creates wireframe geometry from the edges of a 3D solid, surface, mesh, region, or subobject.

**System Variables**

**ELEVATION**
Stores the current elevation of new objects relative to the current UCS.

**FACETRES**
Adjusts the smoothness of shaded and rendered objects and objects with hidden lines removed.

**ISOLINES**
Specifies the number of contour lines per surface on objects.

**Add 3D Thickness to Objects**

Use the thickness property to give objects a 3D appearance.

The 3D thickness of an object is the distance that object is extended, or thickened, above or below its location in space. Positive thickness extrudes upward in the positive Z direction; negative thickness extrudes downward (negative Z). Zero (0) thickness means that there is no 3D thickening of the object.
The orientation of the UCS when the object was created determines the Z direction. Objects with a non-zero thickness can be shaded and can hide other objects behind them.

The thickness property changes the appearance of the following types of objects:

- 2D solids
- Arcs
- Circles
- Lines
- Polylines (including spline-fit polylines, rectangles, polygons, boundaries, and donuts)
- Text (only if created as a single-line text object using an SHX font)
- Traces
- Points

Modifying the thickness property of other types of objects does not affect their appearance.

You can set the default thickness property for new objects you create by setting the THICKNESS system variable. For existing objects, change the thickness property on the Properties Inspector palette.

The 3D thickness is applied uniformly to an object; a single object cannot have different thicknesses.

You might need to change the 3D viewpoint to see the effect of thickness on an object.

**NOTE** Although the THICKNESS variable sets an extruded thickness for new 2D objects, those objects continue to be 2D objects. The THICKEN command adds volume to a surface object, converting it to a 3D solid.
See also:

- Create Solids and Surfaces from Lines and Curves on page 446

Quick Reference

Commands

ELEV
Sets elevation and extrusion thickness of new objects.

PROPERTIES
Controls properties of existing objects.

THICKEN
Converts a surface into a 3D solid with a specified thickness.

System Variables

BACKZ
Stores the back clipping plane offset from the target plane for the current viewport, in drawing units.

FRONTZ
Stores the front clipping plane offset from the target plane for the current viewport, in drawing units.

THICKNESS
Sets the current 3D thickness.

VIEWMODE
Stores the View mode for the current viewport.
Modify 3D Models

Change a 3D solid, surface or mesh with grips and gizmos, editing commands, properties, and by modifying its component subobjects.

Overview of Modifying 3D Objects

3D modeling tools range from entering precise measurements in the Properties Inspector palette, to more free-form methods such as grip and gizmo editing. Some methods are specific to 3D solids, surfaces or meshes. Other methods are shared.

Convert to Other Object Types

In many cases, you can convert from one object type to another to take advantage of specific editing capabilities.

For example you can convert selected surfaces, solids, and legacy mesh types to mesh objects so that you can take advantage of smoothing and modeling capabilities.
Similarly, you can convert mesh to 3D solids and surfaces to accomplish some composite object modeling tasks that are available only for those objects. Conversion is often offered as a choice when you start activities that are available only for solids and surfaces.

View Your Model from All Angles

When you work with any 3D object, you can easily make changes that are not accurately reflected in the current view. To ensure that your modifications
conform to your expectations, make sure you understand and use the following:

- **Manipulate the 3D workplane (UCS).** To understand how your model is projected in 3D space, learn how to use the X, Y, and Z axes. For more information, see Specify Workplanes in 3D (UCS).

- **Rotate the view to display the model from different viewpoints.** Several navigation tools, including 3D Orbit and the ViewCube tool, are available to help you rotate around your workspace. For more information, see Use Viewing Tools on page 107.

- **Display multiple viewports.** Set up two or more viewports with different viewing angles and visual styles. When you make a change in one viewport, you can see its impact from several viewpoints at the same time. For more information, see Display Multiple Views in Model Space on page 121.

See also:

- Specify Workplanes in 3D (UCS) on page 214
- Use Viewing Tools on page 107
- Display Multiple Views in Model Space on page 121

### Use Gizmos to Modify Objects

Use gizmos to move, rotate, or scale objects and subobjects in a 3D view.

### Overview of Using Gizmos

Gizmos help you move, rotate, or scale a set of objects along a 3D axis or plane.

There are three types of gizmos:

- **3D Move gizmo.** Relocates selected objects along an axis or plane.
- **3D Rotate gizmo.** Rotates selected objects about a specified axis.
- **3D Scale gizmo.** Scales selected objects along a specified plane or axis, or uniformly along all 3 axes.
By default, gizmos are displayed automatically when you select an object or subobject in a view that has a 3D visual style. Because they constrain modifications along specific planes or axes, gizmos help ensure more predictable results.

You can specify which gizmos are displayed when an object is selected, or you can suppress their display.

**Quick Reference**

**Commands**

**3DMOVE**

In a 3D view, displays the 3D Move gizmo to aid in moving 3D objects a specified distance in a specified direction.

**3DROTATE**

In a 3D view, displays the 3D Rotate gizmo to aid in revolving 3D objects around a base point.

**3DScale**

In a 3D view, displays the 3D Scale gizmo to aid in resizing 3D objects.

**System Variables**

**DEFAULTGIZMO**

Sets the 3D Move, 3D Rotate, or 3D Scale gizmo as the default during subobject selection.

**GRIPSUBOBJMODE**

Controls whether grips are automatically made hot when subobjects are selected.
GTAUTO
Controls whether 3D gizmos are automatically displayed when you select objects before you start a command in a viewport with a 3D visual style.

GTDEFAULT
Controls whether the 3D Move, 3D Rotate, or 3D Scale operation starts automatically when you start the MOVE, ROTATE, or SCALE command in a viewport with a 3D visual style.

GTLOCATION
Controls the initial location of the 3D Move, 3D Rotate, or 3D Scale gizmo when you select objects before you start a command in a viewport with a 3D visual style.

Use the Gizmos
Gizmos help move, rotate, and scale 3D objects and subobjects.

Display the Gizmos
Gizmos are available only in 3D views that are set to use a 3D visual style such as Hidden. You can set the gizmo to be displayed automatically when you select a 3D object or subobject. Gizmos are also displayed during the 3DMOVE, 3DROTATE, and 3DScale commands.

If the visual style is set to 2D Wireframe, entering 3DMOVE, 3DROTATE, or 3DScale automatically converts the visual style to 3D Wireframe.

By default, the gizmo is initially placed in the center of the selection set. However, you can relocate it anywhere in 3D space. The center box (or base grip) of the gizmo sets the base point for the modification. This behavior is equivalent to temporarily changing the position of the UCS as you move or rotate the selected objects. The axis handles on the gizmo constrain the movement or rotation to an axis or plane.
For best results, use object snaps to locate the grip center box.

Switch Between the Gizmos

Whenever you select an object in a 3D view, the default gizmo is displayed. You can select a different default on the ribbon, or change the value of the DEFAULTGIZMO system variable. You can also suppress the display of gizmos when objects are selected.

After the gizmo is active, you can also switch to a different type of gizmo. The switching behavior differs, depending on when you select the objects:

- **Select objects first.** If a gizmo operation is in progress, you can press the Spacebar repeatedly to cycle through the other gizmo types. When you switch gizmos this way, the gizmo activity is constrained to the originally selected axis or plane.
  During a gizmo operation, you can also select a different gizmo type on the shortcut menu.

- **Run the command first.** When you start the 3D Move, 3D Rotate, or 3D Scale operation before selecting objects, the gizmo is placed at the center of the selection set. Use the Relocate Gizmo option on the shortcut menu to relocate the gizmo anywhere in 3D space. You can also choose a different type of gizmo on the shortcut menu.

Change the Gizmo Settings

The following settings affect the display of gizmos:

- **Default gizmo.** The DEFAULTGIZMO system variable specifies which gizmo is displayed by default when an object is selected in a view with a 3D visual style. You can turn off display of the gizmo. This setting is also available on the status bar.

- **Default location.** The GTLOCATION system variable sets the default location of the gizmo. The gizmo can be displayed at the center of the selection set (default), or it can be positioned at the 0,0,0 coordinates of the current UCS.

- **Automatic display.** The GTAUTO system variable sets whether gizmos are displayed automatically whenever you select objects in a 3D view that is set to a 3D visual style (default). If you turn off this system variable, the grips are not displayed until the gizmos are active.

- **Conversion of move, rotate, and scale operations from 2D to 3D.** Turn on the GTDEFAULT system variable to start the 3DMOVE, 3DROTATE, or
3DSCALE command automatically when the MOVE, ROTATE, or SCALE command is started in a 3D view. This system variable is turned off by default.

- **Active status of subobject grips.** If you select a subobject on page 1030, the GRIPSUBOBJMODE system variable sets whether the subobject grips are active immediately. Setting subobject grips to be active upon selection helps you modify groups of mesh subobjects without selecting them again.

### Quick Reference

#### Commands

**3DMOVE**

In a 3D view, displays the 3D Move gizmo to aid in moving 3D objects a specified distance in a specified direction.

**3DROTATE**

In a 3D view, displays the 3D Rotate gizmo to aid in revolving 3D objects around a base point.

**3DScale**

In a 3D view, displays the 3D Scale gizmo to aid in resizing 3D objects.

#### System Variables

**DEFAULTGIZMO**

Sets the 3D Move, 3D Rotate, or 3D Scale gizmo as the default during subobject selection.

**GRIPSUBOBJMODE**

Controls whether grips are automatically made hot when subobjects are selected.

**GTAUTO**

Controls whether 3D gizmos are automatically displayed when you select objects before you start a command in a viewport with a 3D visual style.

**GTDEFAULT**

Controls whether the 3D Move, 3D Rotate, or 3D Scale operation starts automatically when you start the MOVE, ROTATE, or SCALE command in a viewport with a 3D visual style.
GTLOCATION

Controls the initial location of the 3D Move, 3D Rotate, or 3D Scale gizmo when you select objects before you start a command in a viewport with a 3D visual style.

Move 3D Objects

Move a selection set of objects and subobjects freely or constrain the movement to an axis or plane.

To move 3D objects and subobjects, click and drag the gizmo anywhere in 3D space. This location (indicated by the center box [or base grip] of the gizmo) sets the base point for the movement and temporarily changes the position of the UCS while you move the selected objects.

To move the objects freely, drag outside the gizmo or specify the axis or plane to which you will constrain the movement.

Constrain the Movement to an Axis

You can use the Move gizmo to constrain the movement to an axis. As the cursor hovers over an axis handle on the gizmo, a vector aligned with the axis is displayed, and the specified axis turns yellow. Click the axis handle.
As you drag the cursor, movement of the selected objects and subobjects is constrained to the highlighted axis. You can click or enter a value to specify the distance of the move from the base point. If you enter a value, the movement direction of the object follows the initial direction of the cursor movement.

Constrain the Movement to a Plane

You can use the Move gizmo to constrain the movement to a plane. Each plane is identified by a rectangle that extends from the respective axis handles. You can specify the plane of movement by moving the cursor over the rectangle. When the rectangle turns yellow, click it.

As you drag the cursor, the selected objects and subobjects move only along the highlighted plane. Click or enter a value to specify the distance of the move from the base point.
Quick Reference

Commands
3DMOVE

In a 3D view, displays the 3D Move gizmo to aid in moving 3D objects a specified distance in a specified direction.

System Variables
DEFAULTGIZMO

Sets the 3D Move, 3D Rotate, or 3D Scale gizmo as the default during subobject selection.

Rotate 3D Objects

Constrain the rotation of 3D objects and subobjects to an axis.

After you select the objects and subobjects that you want to rotate, the gizmo is located at the center of the selection set. This location is indicated by the center box (or base grip) of the gizmo. It sets the base point for the movement and temporarily changes the position of the UCS while you rotate the selected objects.
You then rotate the objects freely by dragging outside the gizmo. You can also specify an axis about which to constrain the rotation.

If you want to realign the center of rotation, you can relocate the gizmo by using the Relocate Gizmo option on the shortcut menu.

**Constrain the Rotation to an Axis**

You can constrain the rotation to a specified axis. As you move the cursor over the rotation paths on the 3D rotate gizmo, a vector line representing the axis of rotation is displayed. Specify an axis of rotation by clicking the rotation path when it turns yellow.

When you drag the cursor, the selected objects and subobjects rotate about the base point along the specified axis. The gizmo displays the degree of rotation from the original position of the object as the object moves. You can click or enter a value to specify the angle of the rotation.
Quick Reference

Commands

3DROTATE
In a 3D view, displays the 3D Rotate gizmo to aid in revolving 3D objects around a base point.

System Variables

DEFAULTGIZMO
Sets the 3D Move, 3D Rotate, or 3D Scale gizmo as the default during subobject selection.

Scale 3D Objects

Change the size of 3D objects uniformly or along a specified axis or plane.

After you select the objects and subobjects to scale, constrain the object scaling by clicking the gizmo axis, plane, or the portion of the gizmo between all three axes.

NOTE Non-uniform scaling (along an axis or a plane) is only available for meshes, it is not available for solids and surfaces.

Scale a 3D Object Along an Axis

Constrain mesh object scaling to a specified axis. As you move the cursor over the axes on the 3D Scale gizmo, a vector line representing the axis of scale is displayed. Specify an axis of scale by clicking the axis when it turns yellow.
When you drag the cursor, the selected objects and subobjects are resized along the specified axis. Click or enter a value to specify the scale from the selected base point.

**Scale a 3D Object Along a Plane**

Constrain the mesh object scaling to a specified plane. Each plane is identified by a bar that extends from the outer ends of the respective axis handles. Specify the plane of scale by moving the cursor over one of the bars. When the bar turns yellow, click it.
As you drag the cursor, the selected objects and subobjects are scaled only along the highlighted plane. Click or enter a value to specify the scale from the selected base point.

**Scale a 3D Object Uniformly**

Scale solid, surface, and mesh objects uniformly along all axes. As you move the cursor toward the center point of the gizmo, a highlighted triangular area indicates that you can click to scale the selected objects and subobjects along all three axes.
As you drag the cursor, the selected objects and subobjects are scaled uniformly. Click or enter a value to specify the scale from the selected base point.

Quick Reference

Commands
3DSCALE
In a 3D view, displays the 3D Scale gizmo to aid in resizing 3D objects.

System Variables
DEFAULTGIZMO
Sets the 3D Move, 3D Rotate, or 3D Scale gizmo as the default during subobject selection.

Use Grips to Modify Solids and Surfaces
Use grips to change the shape and size of solids and surfaces.

Use 3D Subobject Grips
Select faces, edges, and vertices on 3D objects.
A subobject is a face, edge or vertex of a solid, surface, or mesh object.
Select Subobjects

To select a face, edge, or vertex of a 3D object, press and hold Ctrl as you select the object. (If you have set a subobject filter, you do not need to hold Ctrl first.)

Selected subobjects display different types of grips, depending on the subobject type.

You can select one or more subobjects on any number of 3D objects. The selection set can include more than one type of subobject. Press and hold Ctrl to select subobjects at the selection prompts of the MOVE, ROTATE, SCALE, and ERASE commands.

You can remove an item from the selection set by pressing and holding Shift and selecting it again.

Select Subobjects on Composite 3D Solids

Press and hold Ctrl to select faces, edges, and vertices on composite solids. If the History property of the composite solid is set to Record (On), the first “pick” might select the history subobject. (The history subobject is the portion of the original object that was removed during the union, subtract, or intersect operation.) Continue to hold Ctrl and pick again to select a face, edge, or vertex on the original form.
If you set a subobject selection filter on page 564, you can select the face, edge, or vertex by clicking it once.

**Quick Reference**

**Commands**

**DSETTINGS**
Sets grid and snap, polar and object snap tracking, object snap modes, and Dynamic Input.

**ERASE**
Removes objects from a drawing.

**MOVE**
Moves objects a specified distance in a specified direction.

**ROTATE**
Rotates objects around a base point.

**SCALE**
Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling.

**SELECT**
Places selected objects in the Previous selection set.

**SOLIDEDIT**
Edits faces and edges of 3D solid objects.

**System Variables**

**GRIPHOVER**
Controls the fill color of an unselected grip when the cursor pauses over it.
GRIPOBJLIMIT
Suppresses the display of grips when the selection set includes more than the specified number of objects.

GRIPS
Controls the display of grips on selected objects.

GRIPSIZE
Sets the size of the grip box in pixels.

GRIPSUBOBJMODE
Controls whether grips are automatically made hot when subobjects are selected.

LEGACYCTRLPICK

SUBOBJSELECTIONMODE
Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

**Cycle Through and Filter Subobjects**
Filter and select faces, edges, and vertices on 3D objects.

A *subobject* is a face, edge or vertex of a solid, surface, or mesh object.

**Cycle Through Multiple Subobjects**
In 3D views, some objects or subobjects might be hidden behind others. You can press Ctrl+Spacebar to cycle through the hidden subobjects until the object you want to select is highlighted.

For example, when you select faces on a box, the face in the foreground is detected first. To select a hidden face, press the Spacebar (with Ctrl still pressed). Release the Spacebar and click to select the face.
Turn on the Subobject Selection Filter

Selecting a specific type of subobject can be difficult on complex objects, such as meshes. You can limit the selection to a face, edge, vertex, or history subobject by setting a subobject selection filter.

When a subobject selection filter is on, you do not need to press and hold Ctrl to select the face, edge, or vertex of a 3D model. However, you need to turn off the filter to select the entire object. The current subobject filter setting is stored in the SUBOBJSELECTIONMODE system variable.

When a subobject filter is turned on, the following icons are displayed near the cursor:

- Vertex filtering is on
- Edge filtering is on
- Face filtering is on
- History subobject filtering is on
- Subobject filtering is off

Quick Reference

Commands

ERASE

Removes objects from a drawing.
MOVE
Moves objects a specified distance in a specified direction.

ROTATE
Rotates objects around a base point.

SCALE
Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling.

SELECT
Places selected objects in the Previous selection set.

SOLIEDIT
Edits faces and edges of 3D solid objects.

**System Variables**

**GRIPHOVER**
Controls the fill color of an unselected grip when the cursor pauses over it.

**GRIPOBJLIMIT**
Suppresses the display of grips when the selection set includes more than the specified number of objects.

**GRIPS**
Controls the display of grips on selected objects.

**GRIPSIZE**
Sets the size of the grip box in pixels.

**GRIPSUBOBJMODE**
Controls whether grips are automatically made hot when subobjects are selected.

**LEGACYCTRLPICK**

**SUBOBJSELECTIONMODE**
Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.
Use Grips to Edit 3D Solids and Surfaces

Use grips to change the size and shape of some individual solids and surfaces.

The method you use to manipulate the 3D solid or surface depends on the type of object and the method used to create it.

**NOTE** For mesh objects, only the center grip is displayed. However, you can edit mesh objects with the 3D Move, Rotate, or Scale gizmos.

**Primitive Solid Forms and Polysolids**

You can drag grips to change the shape and size of primitive solids and polysolids. For example, you can change the height and base radius of a cone without losing the overall cone shape. Drag the top radius grips to transform the cone to a flat-topped, frustum cone.

![Diagram of a cone with grips](image)

**Extruded Solids and Surfaces**

You can convert 2D objects to solids and surfaces with the EXTRUDE command. When selected, extruded solids and surfaces display grips on their profiles. A profile is the original outline that defines the shape of the extruded solid or surface. Drag profile grips to modify the overall shape of the object.

If the extrusion was created along a sweep path, the path can be manipulated with grips. If a path was not used, you can modify the height of the object using a grip at the top of the extruded solid or surface.

**Swept Solids and Surfaces**

Swept solids and surfaces display grips on the swept profile as well as on the sweep path. You can drag these grips to modify the solid or surface.
When you click and drag a grip on the profile, the changes are constrained to the plane of the profile curve.

**Lofted Solids and Surfaces**

Depending on how a lofted solid or surface was created, the solid or surface displays grips on the following, defining lines or curves:

- Cross section
- Path

Drag grips on any of the defining lines or curves to modify the shape. If the lofted object contains a path, you can only edit the portion of the path that is between the first and last cross sections.

You cannot use grips to modify lofted solids or surfaces that are created with guide curves.

**Revolved Solids and Surfaces**

Revolved solids and surfaces display grips on the revolved profile at the start of the revolved solid or surface. You can use these grips to modify the profile of the solid of surface.
A grip is also displayed at the axis of revolution endpoint. You can relocate the axis of revolution by dragging the grip to another location.

See also:
- Create Solids on page 461
- Create a Solid Cone on page 467

Quick Reference

Commands
EXTRUDE
Creates a 3D solid or surface by extending the dimensions of an object.

SELECT
Places selected objects in the Previous selection set.

System Variables
GRIPHOVER
Controls the fill color of an unselected grip when the cursor pauses over it.

GRIPOBJLIMIT
Suppresses the display of grips when the selection set includes more than the specified number of objects.

GRIPS
Controls the display of grips on selected objects.
GRIPSIZETE

Sets the size of the grip box in pixels.

GRIPSUBOBJMODE

Controls whether grips are automatically made hot when subobjects are selected.

**Modify 3D Subobjects**

Modify the shape of a 3D solid or surface by editing its subobjects (faces, edges, and vertices).

**Move, Rotate, and Scale 3D Subobjects**

Move, rotate, and scale individual subobjects on 3D solids and surfaces.

Use the same methods to modify a face, edge, or vertex that you use to modify the entire object:

- Drag grips
- Use gizmos (3DMOVE, 3DROTATE, and 3DScale)
- Enter object editing commands (MOVE, ROTATE, and SCALE)

When you move, rotate, or scale a subobject, the subobject is modified in a way that maintains the integrity of the 3D solid or surface. For example, when you drag an edge to move it, the adjacent faces are adjusted so that they remain adjacent to the edge.

Several results are possible when you modify a solid or surface. When you move, rotate, or scale subobjects, you can press Ctrl one or more times as you drag to cycle through modification options.

The following illustration shows the modification options for moving a face.
Move, Rotate, and Scale Subobjects on Composite Solids

When you modify composite solids, the effect of the edits depends on the current setting of the History property.

- To modify subobjects of each history component separately, the History property must be set to Record (On).
- To modify subobjects of the combined composite solid as a whole, the History property must be set to None (Off).

Rules and Limitations When Moving, Rotating, and Scaling Subobjects

You can only move, rotate, and scale subobjects on 3D solids if the operation maintains the integrity of the solid. The following rules and limitations apply to moving, rotating, and scaling subobjects:

- When you use grips to modify subobjects, grips are not displayed on the subobjects that cannot be moved, rotated, or scaled.
- In most cases, you can move, rotate, and scale both planar and non-planar faces.
- You can only modify an edge that is a straight line and that has at least one planar adjacent face. The planes of the adjacent planar faces are adjusted to contain the modified edge.
- You cannot move, rotate, or scale edges (or their vertices) that are imprinted inside faces.
- You can only modify a vertex if it has at least one planar adjacent face. The planes of the adjacent planar faces are adjusted to contain the modified vertex.
- When you drag a subobject, the final result might be different than the preview displayed during the modification. This result occurs when the solid geometry is adjusted in order to maintain its topology. In some cases,
the modification is not possible because it changes the topology of the solid too severely.

- If the modification causes spline surfaces to be extended, the operation is often unsuccessful.

- You cannot move, rotate, or scale non-manifold edges (edges that are shared by more than two faces) or non-manifold vertices. Also, if some non-manifold edges or vertices are present near faces, edges, and vertices that you modify, the operation might not be possible.

See also:

- Modify Mesh Objects on page 610

Quick Reference

Commands

3DMOVE

In a 3D view, displays the 3D Move gizmo to aid in moving 3D objects a specified distance in a specified direction.

3DROTATE

In a 3D view, displays the 3D Rotate gizmo to aid in revolving 3D objects around a base point.

3DScale

In a 3D view, displays the 3D Scale gizmo to aid in resizing 3D objects.

MOVE

Moves objects a specified distance in a specified direction.

ROTAte

Rotates objects around a base point.

SCALE

Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling.
Modify Faces on 3D Objects

Modify individual faces on 3D objects using a variety of methods.

Move, Rotate, and Scale Faces on 3D Solids and Surfaces

Modify the location, rotation, and size of faces on a 3D solids and surfaces.

Use the MOVE, ROTATE, and SCALE commands to modify faces just as you would with any other object. Press and hold Ctrl while you select a face on a solid.

If you move, rotate, or scale a face on a 3D solid primitive, the solid primitive’s history is removed. The solid is no longer a true primitive and cannot be manipulated using grips or the Properties Inspector palette.

Face Modification Options

As you drag a face, press Ctrl to cycle through modification options.

- **Maintain shape of face, modify adjacent faces.** When you move or rotate a face without pressing Ctrl, the shape and size of the face is maintained. However, the planes of adjacent faces might change.

- **Modify shape of face, retain edges.** When you move or rotate a face and press and release Ctrl once while dragging, the size of the face is modified within the boundary, or footprint, of the adjacent faces.

- **Modify face, triangulate adjacent faces.** When you move or rotate a face and press and release Ctrl twice while dragging, the size and shape of the face is maintained. (This behavior is the same as if you had not pressed
Ctrl). However, the adjacent planar faces are triangulated (divided into two or more planar triangular faces), if necessary.

If you press and release Ctrl a third time, the modification returns to the first option, as if you had not pressed Ctrl.

Quick Reference

Commands

MOVE
Moves objects a specified distance in a specified direction.

ROTATE
Rotates objects around a base point.

SCALE
Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling.

Copy, Delete, and Color Faces on 3D Solids

Copy, remove, or change the color of faces on 3D solid objects.

Copy a Face

You can duplicate the face of a 3D solid object using the copy option of the SOLIDEDIT command. The selected faces are copied as regions or bodies.

Delete a Face

If you specify two points, the first point is used as a base point and a single copy is placed relative to the base point. If you specify a single point and press
Enter, the original selection point is used as a base point. The next point is the point of displacement.

If a face is surrounded by coplanar faces, you can delete it using the following methods:

- Select the face and press Delete.
- Select the face and enter `erase`.
- Use the Delete option of the SOLIDEDIT command.

**Color a Faces**

You can modify the color of a face on a 3D solid by selecting the face and then changing the Color property in the Properties Inspector palette.

**See also:**

- Modify Mesh Faces on page 622

**Quick Reference**

**Commands**

- PROPERTIES
  Controls properties of existing objects.
- SOLIDEDIT
  Edits faces and edges of 3D solid objects.

**Modify Edges on 3D Objects**

You can select and modify edges on a 3D solid or surface.
Move, Rotate, and Scale Edges

Move, rotate, and scale the edges on 3D solids and surfaces using grips, gizmos, and commands.

You can use MOVE, ROTATE, and SCALE to modify edges on 3D solids and surfaces just as you can for any other object. Press and hold Ctrl to select the edge.

If you move, rotate, or scale an edge on a 3D solid primitive, the history of the solid primitive is removed. The solid is no longer a true primitive and cannot be manipulated using grips and the Properties Inspector.

Edges on regions can be selected, but do not display grips. These edges can also be moved, rotated, and scaled.

Edge Modification Options

As you drag an edge, press Ctrl to cycle through modification options.
- **Maintain length of edge.** When you move, rotate, or scale an edge without pressing Ctrl, the shared length of the edge and its vertices is maintained. However, the planes of the adjacent faces adjacent might be changed.

- **Change the length of the edge.** When you move, rotate, or scale an edge and press and release Ctrl once while dragging, the edge is modified without its vertices. The surfaces of the adjacent faces are maintained, but the length of the modified edge might change.

- **Triangulate adjacent faces.** When you move, rotate, or scale an edge and press and release Ctrl twice while dragging, the edge and its vertices are modified. (This behavior is the same as if you had not pressed Ctrl). However, if the adjacent faces are no longer planar, they are triangulated (divided into two or more planar triangular faces).

If you press and release Ctrl a third time, the modification returns to the first option, as if you had not pressed Ctrl.

**Delete Edges**

You can also delete edges that completely divide two coplanar faces using one of the following methods:

- Select the edge and press Delete.
- Select the edge and enter the ERASE command.

**Fillet and Chamfer 3D Solids**

Add rounds and bevels to 3D solids using FILLETEDGE and CHAMFEREDGE.
Use the fillet and chamfer grips to modify the fillet radius or the chamfer distance. The default fillet radius is set by the FILLETRAD3D system variable.

**Color Edges**

You can modify the color of an edge on a 3D object by selecting the edge and changing the Color property in the Properties Inspector.

**Copy Edges**

You can copy individual edges on a 3D solid object. Edges are copied as lines, arcs, circles, ellipses, or splines.

If you specify two points, the first point is used as a base point and a single copy is placed relative to the base point. If you specify a single point, and then press Enter, the original selection point is used as a base point. The next point is used as a point of displacement.

See also:
- Move, Rotate, and Scale 3D Subobjects on page 570
- Use Gizmos to Modify Objects on page 549
- Modify Objects on page 311
- Move or Rotate Objects on page 321
Quick Reference

Commands
EDGE
Changes the visibility of 3D face edges.

CHAMFER
Bevels the edges of objects.

FILLET
Rounds and fillets the edges of objects.

MOVE
Moves objects a specified distance in a specified direction.

PROPERTIES
Controls properties of existing objects.

ROTATE
Rotates objects around a base point.

SCALE
Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling.

SOLIDEEDIT
Edits faces and edges of 3D solid objects.

System Variables

FILLETRAD3D
Stores the current fillet radius for 3D objects.

Modify Vertices on 3D Objects
Move, rotate, scale, or drag the vertices of 3D solids and surfaces.
You can modify the form of a 3D solid or surface by modifying one or more vertices. Use grips and gizmos, or run the MOVE, ROTATE, or SCALE command. When you scale or rotate vertices, you must select two or more vertices to see a change in the object. Clicking and dragging a vertex “stretches” the 3D object.

If you move, rotate, or scale one or more vertices on a 3D solid primitive, the solid primitive history is removed. The solid is no longer a true primitive and cannot be modified using grips and the Properties Inspector palette.

**Vertex Modification Options**

As you drag a vertex, press Ctrl to cycle through modification options.

- **Triangulate adjacent faces.** When you move, rotate, or scale a vertex without pressing Ctrl, some adjacent planar faces may be triangulated (divided into two or more planar triangular faces).
■ **Modify some adjacent faces without triangulation.** When you move, rotate, or scale a vertex and press and release Ctrl once, some adjacent planar faces might be adjusted.

If you press and release Ctrl a second time, the modification returns to the first option, as if you had not pressed Ctrl.

**Delete a Vertex**

You can delete a vertex that connects two parallel edges that are collinear and do not intersect on any other edges.

**See also:**

■ *Move, Rotate, and Scale 3D Subobjects* on page 570
■ *Use Gizmos to Modify Objects* on page 549
■ *Modify Objects* on page 311
■ *Overview of Modifying Meshes* on page 610

**Quick Reference**

**Commands**

**MOVE**

Moves objects a specified distance in a specified direction.

**ROTATE**

Rotates objects around a base point.

**SCALE**

Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling.

**SOLIDEDIT**

Edits faces and edges of 3D solid objects.

**Work with Complex 3D Solids and Surfaces**

Modify composite solids that are created by a union, subtract, intersect, fillet, or chamfer process.
Display Original Forms of Composite Solids

By default, 3D composite objects retain a history that displays an editable image of their original component forms.

Retain the History of the Composite Components

After you create a composite object, you can modify the shape of the new object by modifying a highlighted wireframe image of its original components. If the Show History property is Yes (On), wireframes of the original forms—including forms that have been removed—are displayed in a dimmed state. (The SHOWHIST system variable also controls this setting.)

To retain a history of the original parts of composite solids, the History property must be set to Record (On) in the Properties Inspector palette when the composite operation occurs. (You can also use the SOLIDHIST system variable to set this property.)

Display and Remove the History to Modify the Composite

When you modify the composite object, you can display the history. Then use the grips on the history subobject to modify the object. For more information about using grips with composite solids, see Modify Composite Solids and Surfaces on page 583.

You can remove the history of a selected composite object by changing its History setting to None, or by entering the BREP command. After a history has been removed, you can no longer select and modify the original, removed, components of the solid. You can restart history retention for the solid by changing its History setting back to Record.
Removing a composite history is useful when you work with complex composite solids. After you create the initial complex form, set History to None (Off) to remove the history. Then reset the value to Record (On). With this process, you can create a complex composite object, and then reset it to serve as a base form for additional composite operations.

See also:

- Modify Composite Solids and Surfaces on page 583

Quick Reference

Commands

BREP

Removes the history from 3D solids and composite solids, and associativity from surfaces.

System Variables

SHOWHIST

Controls the Show History property for solids in a drawing.

SOLIDHIST

Controls the default history property setting for solid objects.

Modify Composite Solids and Surfaces

Modify the entire form of a composite 3D object or the original forms that make up the composite.
You can move, scale, or rotate a selected composite object using grips or gizmos.

**Modify Original Components of Composites**

When the History property is set to Record (On), press the Ctrl key to display any original forms that were removed during a union, subtract, or intersect operation. If the original, removed form was a solid primitive, you can drag the displayed grips to change its shape and size. As a result, the composite object is modified.

If the selected individual form does not contain its history, you can move, rotate, scale, or delete the form.

**Modify Complex Composites**

A composite object might be made up of other composite objects. You can select the history images of composite objects by holding down the Ctrl key as you click the forms. (For best results, set the subobject selection filter to Solid History.)

You can also change the size and shape of composite objects by clicking and dragging grips on individual faces, edges, and vertices. For more information, see [Modify 3D Subobjects](#) on page 570.
Separate Discrete Objects Combined with a Union

If you have combined discrete 3D solids or surfaces using a union operation, you can separate them into their original components. (Use the Separate option of the SOLIDEDIT command.) Composite objects cannot overlap or share a common area or volume to be separated.

After separation, the individual solids retain their original layers and colors. All nested 3D solid objects are restored to their simplest forms.

See also:
- Edit Objects with Grips on page 313
- Overview of Modifying Meshes on page 610

Quick Reference

Commands
SOLIDEDIT
Edits faces and edges of 3D solid objects.

System Variables
LEGACYCTRLPICK

Shell and Remove Redundancies in 3D Objects

Convert 3D solids to shells and remove redundant lines and edges.

Shell 3D Solids

Convert a 3D solid to a hollow wall, or shell.

When you can convert a solid object to a shell, new faces are created by offsetting existing faces inside or outside their original positions.
Continuously tangent faces are treated as a single face when they are offset.

**Quick Reference**

**Commands**

SOLIEDIT

Edits faces and edges of 3D solid objects.

**Clean and Check 3D Solids**

Remove redundant faces, edges, and vertices from a 3D solid, and verify whether the 3D solid is valid.

You can remove redundant edges or vertices that share the same surface or vertex definition. This operation merges adjacent faces and deletes all redundant edges, including imprinted and unused edges.

True 3D solid objects have editable properties, volume, and mass that are not shared by objects created with thickness or closed surfaces. You can check whether an object is a valid 3D solid by verifying whether it is listed as “3D Solid” on the Properties Inspector palette. You can also use SOLIEDIT to verify whether a solid object is a valid 3D solid object.
Quick Reference

Commands

SOLIDEDIT
Edits faces and edges of 3D solid objects.

System Variables

SOLIDCHECK
Turns 3D solid validation on and off for the current session.

Press or Pull Bounded Areas

Create a positive or negative extrusion in the shape of a bounded area.

Press in or pull out bounded, or closed, areas to create 3D holes and positive extrusions.
In combination with imprinted faces, you can form complex shapes using press or pull operations to create extrusions and notches.

**Methods for Press and Pull Modifications**

With the PRESSPULL command, you specify the area to be extruded, and then move the cursor or enter a value to specify the length of the extrusion. The result is a single 3D solid object, often with a composite shape.

**NOTE** If you alternatively use EXTRUDE to extend an existing face on a 3D solid, a separate extruded object is created.

**Types of Objects That Can Be Pressed or Pulled**

You can press or pull several types of bounded areas, including closed objects, areas enclosed by coplanar geometry, the faces of 3D solids, and an imprinted area on the face of a 3D solid. For a complete list of objects that are eligible for a press or pull extrusion, see PRESSPULL.

You cannot taper the pressed or pulled shape as you create it. However, you can achieve the same effect later by modifying the edges of the bounded area.

**Quick Reference**

**Commands**

- **EXTRUDE**
  
  Creates a 3D solid or surface by extending the dimensions of an object.

- **PRESSPULL**
  
  Presses or pulls bounded areas.

**System Variables**

- **IMPLIEDFACE**
  
  Controls the detection of implied faces.

**Add Edges and Faces to Solids**

Add editable faces to 3D solids and surfaces by imprinting other objects, such as arcs and circles.
With the IMPRINT command, you can add a new face to a 3D solid by imprinting a coplanar object that overlaps the selected face. Imprinting provides additional edges that you can modify to reshape the solid object.

For example, if a circle overlaps the face of a box, you can imprint the intersecting curves on the solid.

You can delete or retain the original object as you imprint it.

Objects that can be imprinted on 3D solids include arcs, circles, lines, 2D and 3D polylines, ellipses, splines, regions, bodies, and other 3D solids.

**Edit Imprinted Objects**

You can edit imprinted objects and subobjects in many of the same ways that you can edit other faces. For example, you can Ctrl+click to select a new edge and then drag to change its location.

The following limitations exist for imprinted objects:

- You can move the edges of the imprinted face only within the plane of a face.
- You might be unable to move, rotate, or scale some subobjects.
- Imprinted edges and faces might be lost when some subobjects are moved, rotated, or scaled.

Subobjects with editing limitations include

- Faces with imprinted edges or faces
- Edges or vertices with adjacent faces that contain imprinted edges or faces
Quick Reference

Commands

IMPRINT

Imprints 2D geometry on a 3D solid or surface, creating additional edges on planar faces.

Modify Properties of 3D Solid, Surface, and Mesh

Modify 3D objects by changing their settings in the Properties Inspector palette.

3D solids, surfaces, and meshes, and their subobjects can be modified in the Properties Inspector palette.

Modify Solid Objects by Changing Properties

By changing settings in the Properties Inspector palette, you can modify basic size, height, and shape characteristics of primitive solids. For example, to change a four-sided pyramid that ends in a point to an eight-sided pyramid that ends in a planar surface (pyramid frustum), update the Top Radius and Sides properties.

Set Whether to Retain Compound Object History

With 3D solids that have been recombined to form compound objects, you can choose to retain the history subobject, which represents components that have been removed. The Properties Inspector palette controls the availability and display of these histories. For more information, see Work with Complex 3D Solids and Surfaces on page 581.
Modify Surface Objects by Changing Properties

Surface objects have additional properties that are not found in 3D solid or mesh objects. The properties differ depending on the type of surface (NURBS, blend, patch, network, offset, fillet, chamfer, extend, loft, extrude, sweep, planar, or revolve).

Surfaces include the following information in the Properties Inspector palette:

- **Basic geometric information** - Contains information such as radius for fillet surfaces, offset distance for offset surfaces, and taper angle for extruded surfaces. You can also enter mathematical expressions to control some of these properties.

- **Maintain Associativity** - Displays whether surface is associative or not. Use this property to turn associativity off.

- **Show Associativity** - Turns dependency highlighting on and off if the surface is associated with other surfaces.

- **Edge Continuity and Bulge Magnitude** - Displays for surfaces that join other surfaces.

- **Wireframe Display and U/V Isolines** - Turns the wireframe and U/V Isoline display on and off (for non-NURBS surfaces).

- **CV Hull Display and U/V Isoparms** - Turns the CV Hull and U/V Isoparm display on and off (for NURBS surfaces).

- **Trims** - Reports whether the surface has any trimmed areas and on which edges.
Modify Mesh Objects by Changing Properties

Mesh objects have additional properties that control the level of smoothness and creases. Crease properties of face, edge, and vertex subobjects are also reflected in the Properties Inspector palette.

- **Level of Smoothness.** Smooths or sharpens the edges of a mesh object.

- **Crease Type.** Specifies the presence of a crease (or sharpened edge) and the effect of smoothing. Smoothing does not affect a crease with a value of Always. A crease set to By Level retains its sharpness until the mesh object is smoothed to the specified crease level.

- **Crease Level.** When a crease is set to By Level, indicates the smoothing level at which the crease starts to lose its sharpness.
Modify 3D Subobject Properties

In addition to solids, surfaces, and meshes, you can also use the Properties Inspector palette to modify the properties of individual subobjects, such as faces, edges, and vertices. Different properties are available for different types of subobjects.

In some cases, the application of properties can differ depending on the object type. For example, you can modify the properties of mesh faces, including their color. However, the color appearance of a mesh face might differ from the equivalent color on a 3D solid face. This difference occurs because changing the color of a face modifies the **diffuse color** of the face, but not the **ambient color** (which is derived from the mesh material property). To obtain a closer match between the color of 3D solid and mesh faces, you can add lights and turn off the default lighting (which disables ambient lighting). You can also try assigning a material that has the same ambient and diffuse color. For more information see Create Materials.

See also:
- Work with Complex 3D Solids and Surfaces on page 581
- Create Materials

**Quick Reference**

**Commands**

**PROPERTIES**

Controls properties of existing objects.
**System Variables**

**FILLETRAD**
Stores the current fillet radius for 2D objects.

**FILLETRAD3D**
Stores the current fillet radius for 3D objects.

**LOFTANG1**
Sets the draft angle through the first cross section in a loft operation.

**LOFTANG2**
Sets the draft angle through the last cross section in a loft operation.

**LOFTMAG1**
Sets the magnitude of the draft angle through the first cross section in a loft operation.

**LOFTMAG2**
Sets the magnitude of the draft angle through the last cross section in a loft operation.

**Modify Surfaces**

Reshape surfaces, and then analyze and rebuild the model if necessary to ensure quality and smoothness.

**Overview of Modifying Surfaces**

Fillet, extend, and trim surfaces or modify NURBS surfaces with control vertices. Procedural and NURBS surfaces can be edited with basic editing tools such as trimming, extending and filleting. NURBS surfaces can be modified with these tools as well, but they can also be reshaped by stretching control vertices. When your surface design is complete, use the surface analysis tools to ensure the quality of your model and to rebuild it if necessary.

- **Trim and Untrim Surfaces**
- Extend Surfaces

- Fillet Surfaces
Trim and Untrim Surfaces

Trim and untrim surfaces to meet the edges of other objects.

An important step in the surface modeling workflow is trimming surfaces. You can trim a surface where it meets an intersecting object or you can project geometry onto a surface as a trimming edge.

When a surface is trimmed, the removed surface areas can be replaced with SURFUNTRIM.
NOTE SURFUNTRIM does not restore areas removed by the SURFAUTOTRIM system variable and PROJECTGEOMETRY. It only restores areas trimmed with SURFTRIM.

The Properties Inspector palette indicates if the surface contains any trimmed edges.

**Projecting Geometry onto Surfaces, Solids, and Regions**

Similar to projecting a movie onto a screen, you can project geometry onto 3D solids, surfaces, and regions from different directions to create trimming edges. The PROJECTGEOMETRY command creates a duplicate curve on the object that you can move and edit. You can also trim against 2D curves that do not actually touch the surface, but that appear to intersect the object in the current view.

Use the SURFACEAUTOTRIM system variable to automatically trim a surface when you project geometry onto it.

**Options for Projecting Geometry**

Project geometry from 3 different angles: the Z axis of the current UCS, the current view, or a path between two points.

- **Project to UCS** - Projects the geometry along the positive or negative Z axis of the current UCS.
- **Project to View** - Projects the geometry based on the current view.
- **Project to Two Points** - Projects the geometry along a path between two points.

**Quick Reference**

**Commands**

3DOSNAP

Sets the object snap modes for 3D objects.

PROJECTGEOMETRY

Projects points, lines, or curves onto a 3D solid or surface from different directions.
PROPERTIES
Controls properties of existing objects.

SURFTRIM
Trims portions of a surface where it meets another surface or type of geometry.

SURFUNTRIM
Replaces surface areas removed by the SURFTRIM command.

VISUALSTYLES (-VISUALSTYLES)
Creates and modifies visual styles and applies a visual style to a viewport.

System Variables

3DOSMODE
Controls the settings for the 3D object snaps.

SUBOBJSELECTIONMODE
Filters whether faces, edges, vertices or solid history subobjects are highlighted when you roll over them.

SURFACEAUTOTRIM
Controls whether surfaces are automatically trimmed when you project geometry onto them.

Extend a Surface
Create a new surface by extending it to meet the edge of another object or by specifying an extension length.

There are two types of extend surfaces: merge and append. The merge surface is a continuation of the surface with no seam. The append surface extends the surface by adding a second surface with a seam. Because it creates a seam, append surfaces have continuity and bulge magnitude on page 488 properties.

For both surface types, use the Properties Inspector palette to change the length or to derive the length from a mathematical expression.
Quick Reference

Commands

PROPERTIES
  Controls properties of existing objects.
SURFEXTEND
  Lengthens a surface by a specified distance.

Fillet a Surface

Create a new transition surface that fillets an area between two existing surfaces or regions.

Create a tangent surface between two surfaces or regions with a constant radius profile. The original surfaces will trim to meet the fillet surface.

By default, the fillet surface uses the radius value set in the FILLETRAD3D system variable. Change the radius while you are creating the surface with the radius option or by dragging the fillet grip. Use the Properties Inspector palette to change the fillet radius or to derive the radius using a mathematical expression.

Quick Reference

Commands

SURFFILLET
  Creates a filleted surface between two other surfaces.

PROPERTIES
  Controls properties of existing objects.

System Variables

FILLETRAD3D
  Stores the current fillet radius for 3D objects.
Edit NURBS Surfaces

Change the shape of NURBS surfaces and curves by using the 3D Edit Bar or by editing control vertices.

Use the Control Vertices Edit Bar (3DEDITBAR) to drag and reshape surfaces.
A second way to edit NURBS surfaces is to drag and edit the control vertices directly. Press and hold Shift to select multiple control vertices.

Use CVSHOW to display the control vertices for both NURBS surfaces and curves.

Drag the control vertices to reshape the curve or surface; you can also add or delete control vertices in both the $U$ and $V$ directions.

The typical surface modeling workflow is to:

- Create a model that combines 3D solids, surfaces, and mesh objects.
- Convert the model to procedural surfaces to take advantage of associative modeling.
- Convert the procedural surfaces to NURBS surfaces with CONVTONURBS to take advantage of NURBS editing.
- Check for imperfections and wrinkles with the surface analysis tools.
- If necessary, rebuild the surfaces to restore smoothness with CVREBUILD.
Quick Reference

Commands

3DEDITBAR
Reshapes, scales, and edits the tangency of NURBS surfaces.

CONVTONURBS
Converts 3D solids and surfaces into NURBS surfaces.

CVSHOW
Displays the control vertices for specified NURBS surfaces or curves.

CVADD
Adds control vertices to NURBS surfaces and splines.

CVHIDE
Turns off the display of control vertices for all NURBS surfaces and curves.

CVREBUILD
Rebuilds the shape of NURBS surfaces and curves.

CVREMOVE
Removes control vertices from NURBS surfaces and curves.

System Variables

SURFACEMODELINGMODE
Controls whether surfaces are created as procedural surfaces or NURBS surfaces.

Rebuild NURBS Surfaces and Curves

Reconstruct NURBS surfaces and curves to a specified degree and number of control vertices.

Editing a NURBS surface or curve can create discontinuity and wrinkles. Reconstruct the surface or curve by changing the degree and the number of control vertices. Rebuilding also allows you to delete the original geometry, and, for surfaces only, to replace trimmed areas.
Quick Reference

Commands

CONVTONURBS

CVSHOW
Displays the control vertices for specified NURBS surfaces or curves.

CVADD
Adds control vertices to NURBS surfaces and splines.

CVHIDE
Turns off the display of control vertices for all NURBS surfaces and curves.

CVREBUILD
Rebuilds the shape of NURBS surfaces and curves.

CVREMOVE
Removes control vertices from NURBS surfaces and curves.

Analyze Surfaces

Surface analysis tools check the continuity, curvature and draft angles of surfaces.

Use the surface analysis tools to validate surfaces and curves before manufacturing. Analysis tools include:

- Zebra Analysis on page 606 - Analyzes surface continuity by projecting parallel lines onto the model.
- **Curvature Analysis** on page 608 - Evaluates areas of high and low surface curvature by displaying a color gradient.

- **Draft Analysis** on page 609 - Evaluates whether a model has adequate draft between a part and its mold.
NOTE Analysis tools only work in the 3D visual styles; they will not work in 2D.

Quick Reference

Commands

ANALYSIS CURVATURE
Displays a color gradient onto a surface to evaluate different aspects of its curvature.

ANALYSIS DRAFT
Displays a color gradient onto a 3D model to evaluate whether there is adequate space between a part and its mold.

ANALYSIS OPTIONS
Sets the display options for zebra, curvature, and draft analysis.

ANALYSIS ZEBRA
Projects stripes onto a 3D model to analyze surface continuity.
System Variables

VSACURVATUREHIGH
Sets the value at which a surface displays as green during curvature analysis (ANALYSESCURVATURE).

VSACURVATURELOW
Sets the value at which a surface displays as blue during curvature analysis (ANALYSESCURVATURE).

VSACURVATURETYPE
Controls which type of curvature analysis is used with the (ANALYSESCURVATURE).

VSADRAFANGLEHIGH
Sets the value at which a model displays as green during draft analysis (ANALYSISDRAFT).

VSADRAFANGLELOW
Sets the value at which a model displays as blue during draft analysis (ANALYSISDRAFT).

VSAZEBRACOLOR1
Sets the first color of the zebra stripes displayed during zebra analysis (ANALYSISZEBRA).

VSAZEBRACOLOR2
Sets the second (contrasting) color of the zebra stripes displayed during zebra analysis (ANALYSISZEBRA).

VSAZEBRADIRECTION
Controls whether zebra stripes display horizontally, vertically, or at an angle during zebra analysis (ANALYSISZEBRA).

VSAZEBRASIZE
Controls the width of the zebra stripes displayed during zebra analysis (ANALYSISZEBRA).

VSAZEBRATYPE
Sets the type of zebra display when using zebra analysis (ANALYSISZEBRA).
Analyze Surface Continuity with Zebra Analysis

The zebra analysis tool projects stripes onto a surface so that you can inspect the continuity between surfaces.

Surface continuity is a measure of how smoothly two surfaces flow into each other. A car hood, for example, can be composed of multiple small surfaces that appear to be one because of the smoothness of the surface continuity.

NOTE Analysis tools only work in the 3D visual styles; they will not work in 2D.

How to Interpret the Zebra Stripes

In the seam where two surfaces meet, the way that the zebra stripes align and curve tells you a lot about the smoothness of the join.

■ **G0 Position.** The position of the surface edges is collocated; they touch. But the tangency and curvature do not match. The zebra stripes do not line up.

■ **G1 Tangency.** The position and tangency of surfaces is the same. This indicates G1 (G0 + G1 or position + tangency). The zebra stripes line up, but they veer away from one another at sharp curves.
**G2 Curvature.** The position, tangency, and curvature of the surface edges is the same. This indicates G2 (G0 + G1 + G2 or position + tangency + curvature).

The stripes line up, but they do not veer away from each other at sharp curves (because they share the same curvature). This distinction is subtle and a little harder to discern from G1 continuity.

---

**Quick Reference**

**Commands**

**ANALYSISOPTIONS**

Sets the display options for zebra, curvature, and draft analysis.

**ANALYSISZEBRA**

Projects stripes onto a 3D model to analyze surface continuity.
System Variables

VSAZEBRACOLOR1
Sets the first color of the zebra stripes displayed during zebra analysis (ANALYSISZEBRA).

VSAZEBRACOLOR2
Sets the second (contrasting) color of the zebra stripes displayed during zebra analysis (ANALYSISZEBRA).

VSAZEBRADIRECTION
Controls whether zebra stripes display horizontally, vertically, or at an angle during zebra analysis (ANALYSISZEBRA).

VSAZEBRASIZE
Controls the width of the zebra stripes displayed during zebra analysis (ANALYSISZEBRA).

VSAZEBRATYPE
Sets the type of zebra display when using zebra analysis (ANALYSISZEBRA).

Analyze the Curvature of a NURBS Surface
Displays a color gradient onto surfaces to evaluate areas of high, low and Gaussian curvature.

The color gradient allows you to visualize Gaussian, minimum, maximum, and mean $U$ and $V$ surface curvature. Maximum curvature and a positive Gaussian value display as red, and minimum curvature and a negative Gaussian value display as blue. Positive Gaussian curvature means the surface is shaped like a bowl. Negative Gaussian curvature means the surface is shaped like a saddle as shown in this illustration. Mean curvature and a zero Gaussian value means the surface is flat in at least one direction (planes, cylinders, and cones have zero Gaussian curvature).

The curvature analysis tool ensures that your model stays within a specified range. You can analyze the following on a point on a surface or curve:

NOTE Analysis tools only work in the 3D visual styles; they will not work in 2D.
Quick Reference

Commands

ANALYSESCURVATURE
Displays a color gradient onto a surface to evaluate different aspects of its curvature.

ANALYSISOPTIONS
Sets the display options for zebra, curvature, and draft analysis.

System Variables

VSACURVATUREHIGH
Sets the value at which a surface displays as green during curvature analysis (ANALYSESCURVATURE).

VSACURVATURELOW
Sets the value at which a surface displays as blue during curvature analysis (ANALYSESCURVATURE).

VSACURVATURETYPE
Controls which type of curvature analysis is used with the (ANALYSESCURVATURE).

Use the Draft Analysis Tool

Evaluates if a model has adequate space between a part and its mold.

If you are creating shapes or parts that need to be molded, the draft analysis tool evaluates if their is adequate draft between a part and its mold (based on the pull direction).

NOTE Analysis tools only work in the 3D visual styles; they will not work in 2D.

Quick Reference

Commands

ANALYSISDRAFT
Displays a color gradient onto a 3D model to evaluate whether there is adequate space between a part and its mold.
ANALYSISOPTIONS
Sets the display options for zebra, curvature, and draft analysis.

System Variables
VSADRAFTANGLEHIGH
Sets the value at which a model displays as green during draft analysis (ANALYSISDRAFT).

VSADRAFTANGLELOW
Sets the value at which a model displays as blue during draft analysis (ANALYSISDRAFT).

Modify Mesh Objects
Model mesh objects by changing smoothing levels, refining specific areas, or adding creases.

Overview of Modifying Meshes
Modeling mesh objects differs from modeling 3D solids and surfaces in some important ways.
Mesh objects do not have the mass and volume properties of 3D solids. However, they do offer unique capabilities that enable you to design less angular, more rounded models. Mesh objects are easier to mold and reshape than their solid and surface counterparts.
NOTE The capabilities described in this section apply only to mesh objects created in AutoCAD 2010 and later. They cannot be used with legacy polyface or polygon mesh.

About Mesh Faces

Mesh objects are composed of faces and facets.
Faces are non-overlapping units that—along with their edges and vertices—form the basic editable units of a mesh object. When you move, rotate, and scale individual mesh faces, surrounding faces are stretched and deformed in order to avoid introducing gaps. When gaps occur, you can often close them by smoothing the object or refining individual faces.

About Mesh Facets

Mesh faces have underlying structures, known as facets. The density of the facet grid corresponds to the smoothness of the mesh. As the smoothness level is increased, the density of the underlying facet grid also increases. When you want to confine detailed mesh editing to a smaller area, you can convert facets to editable faces by using refinement.

Unlike faces, facets cannot be individually modified. However, you can make them more visible by modifying the VSLIGHTINGQUALITY system variable.

About Mesh Modeling

You can work with mesh objects in the following ways:

- **Add smoothness.** Increase or decrease smoothness levels to round the overall shape of the model. The underlying density of the mesh facet grid increases as the mesh object smoothness level increases (MESHSMOOTHMORE, MESHSMOOTHLESS).

- **Refine the object to reset the baseline smoothness level.** Refine a mesh object to convert the underlying facet grid to editable faces. Refinement
also resets the lowest level of smoothness that can be applied to the object (MESHREFINE).

- **Refine a face.** Restrict the refinement to a specific mesh face. This method avoids resetting the smoothness baseline.

- **Crease an edge.** Remove the smoothness from specified edges. You can also remove an existing crease (MESHCREASE).

- **Split or merge faces.** Divide an existing face into separate components along a specified path. Merge two or more faces to create a single face (MESH_SPLIT MESH MERGE).

- **Collapse vertices.** Alter the mesh model by collapsing the vertices of adjacent faces to a single point (MESH_COLLAPSE).

- **Spin edges.** Spin the shared edge of adjacent triangular faces to alter the shapes and orientation of the faces (MESH_PIN).

- **Extrude a face.** Extend a specified face by extruding it into 3D space. Unlike 3D solid extrusion, a mesh extrusion does not create a separate object (MESH_EXTRUDE).

- **Repair holes.** Close a gap between faces by selecting the surrounding edges. Holes in mesh objects can prevent you from converting a mesh object to a solid object (MESH_CAP).

### Use Grip Editing with Mesh

Grips, as described in [Use Grips to Edit 3D Solids and Surfaces](#) on page 567, are not available with meshes. However, you can manipulate the entire mesh model or individual subobjects using the following methods:

- **Subobject selection and editing.** Select faces, edges, and vertices the same way you select 3D solid subobjects. Press and hold Ctrl while selecting a subobject. The subobject highlighting indicates what is selected. Press and hold Shift and click again to remove the selection from a subobject. By turning on the Subobject Selection Filter, you can restrict selection to a specific subobject, which you can select without pressing and holding Ctrl. See [Use 3D Subobject Grips](#) on page 561.

- **Gizmo editing.** When you select a mesh object or subobject, the 3D Move, Rotate, or Scale gizmo is displayed automatically. (You can set which gizmo is displayed by default.) Use these gizmos to modify the selection uniformly, or along a specified plane or axis. For more information, see [Use Gizmos to Modify Objects](#) on page 549.
Because dense meshes can be difficult to work with, you can change settings to improve the display and behavior of grips.

- **Set the subobject selection filter to select only faces, edges, or vertices:** Set the DEFAULTGIZMO system variable or use the shortcut menu.

- **Set whether a grip on a face, edge, or vertex is active immediately when you select the subobject:** Set the GRIPSUBOBJMODE system variable.

**See also:**
- Use Grips to Edit 3D Solids and Surfaces on page 567
- Use Gizmos to Modify Objects on page 549

**Quick Reference**

**Commands**

**MESHCAP**
- Creates a mesh face that connects open edges.

**MESHCOLLAPSE**
- Merges the vertices of selected mesh faces or edges.

**MESHCREASE**
- Sharpens the edges of selected mesh subobjects.

**MESHEXTRUDE**
- Extends a mesh face into 3D space.

**MESHMERGE**
- Merges adjacent faces into a single face.

**MESHREFINE**
- Multiplies the number of faces in selected mesh objects or faces.

**MESHSMOOTHLESS**
- Decreases the level of smoothness for mesh objects by one level.

**MESHSMOOTHMORE**
- Increases the level of smoothness for mesh objects by one level.
MESHSPIN
Spins the adjoining edge of two triangular mesh faces.

MESH_SPLIT
Splits a mesh face into two faces.

MESHUNCREASE
Removes the crease from selected mesh faces, edges, or vertices.

System Variables
DEFAULTGIZMO
Sets the 3D Move, 3D Rotate, or 3D Scale gizmo as the default during subobject selection.

GRIPSUBOBJMODE
Controls whether grips are automatically made hot when subobjects are selected.

VSLIGHTINGQUALITY
Sets the lighting quality in the current viewport.

Change Mesh Smoothness Levels

Increase the roundness of mesh objects by increasing the smoothness levels.

Mesh objects are made up of multiple subdivisions, or tessellations, which define the editable faces. Each face consists of underlying facets. When you increase smoothness, you increase the number of facets to provide a smoother, more rounded look.
Increase or Decrease Smoothness

As you work, you can increase and decrease the level of smoothness. The differences are apparent both in the wireframe and conceptual visual styles and in the rendered output.
The lowest level of smoothness, or baseline, is 0. By default, Level 0 has no smoothness. You can increase the smoothness of any mesh object up to the current limits. However, you cannot decrease the smoothness of a mesh object whose level of smoothness is zero.

If you have added creases to a mesh object, the effect of smoothing differs, depending on the crease setting. The effect of creases added to mesh that has no smoothness (Level 0) is not apparent until the mesh is smoothed.

As you edit an object using gizmos or grips, you might create gaps in the mesh object. One way to close the gap is to smooth the object or refine individual subobjects.

**NOTE** Using hardware acceleration might also help resolve this problem. (For more information, see Graphics System.)

**Limit Mesh Density**

Mesh is created at the level of smoothness that you specify. The smoothness can range from None (0), to the default maximum (6), or to a level that you specify. As an object is smoothed, the density of the mesh facet grid also increases. For best results, model mesh objects at lower smoothness levels and increase the smoothness only after modeling is complete.
Dense meshes can result in subobjects that are difficult to select and edit. They can also affect performance. Therefore you might want to set limits that prevent the mesh from becoming too dense.

- **Maximum level of smoothness at which a grid is displayed** (SMOOTHMESHGRID). Displays the effects of modeling without the complexity of the underlying facet grid. The default smoothness level is 3. The tessellation display becomes increasingly dense until the maximum level is exceeded. Beyond that level, the display reverts to the most basic level, even though the smoothing level can continue to increase.

- **Maximum number of faces in a drawing** (SMOOTHMESHMAXFACE). Sets the maximum number of mesh faces that are permitted per mesh object.

- **Maximum level of smoothness** (SMOOTHMESHMAXLEV). Sets the maximum smoothness level permitted for mesh objects.

**Quick Reference**

**Commands**

- MESHSMOOTHLESS
  - Decreases the level of smoothness for mesh objects by one level.

- MESHSMOOTHMORE
  - Increases the level of smoothness for mesh objects by one level.

**Properties**

- Controls properties of existing objects.

**System Variables**

- SMOOTHMESHGRID
  - Sets the maximum level of smoothness at which the underlying mesh facet grid is displayed on 3D mesh objects.

- SMOOTHMESHMAXFACE
  - Sets the maximum number of faces permitted for mesh objects.

- SMOOTHMESHMAXLEV
  - Sets the maximum smoothness level for mesh objects.
**Refine Mesh Objects or Subobjects**

Refine a mesh object or subobject to convert underlying facets to editable faces.

You can refine any mesh that has a level of smoothness of 1 or higher.

**Refine a Mesh Object and Reset the Baseline**

Refining an object increases the number of editable faces by converting the underlying facets to faces. The number of resulting faces depends on the current level of smoothness. Higher smoothness levels result in a higher number of faces after refinement.

In addition to increasing the number of faces, refining a mesh object resets its level of smoothness back to the baseline. Therefore, an object might appear to be smoothed, but its smoothness level can still equal 0 (zero).

**Refine a Mesh Face**

You can refine an entire mesh object as shown in the previous illustration, or select a specific face to refine. A refined face is subdivided into four faces and the surrounding faces are deformed slightly to accommodate the change.
Refining a mesh face does not affect the overall smoothing level of the mesh object. Unlike a refined mesh object, refined faces can be refined again immediately. With mesh face refinement, you can target smaller areas for detailed modeling.

**How Refinement Affects Creases**

A crease that is set to Always retains its sharpness no matter how much you smooth or refine the object. However, the behavior is different when you assign a crease value. If you refine an object or edge that has a crease value, the assigned crease value is lowered by the value of the original level of smoothing. Suppose that you add a crease with a crease value of 4 and then refine a mesh whose level of smoothness is 2. The new crease value is 2.

If a crease is applied before an object is smoothed or refined, the effect is not apparent until after the object is smoothed or refined.
Quick Reference

Commands
MESHREFINE
Multiplies the number of faces in selected mesh objects or faces.

Add Creases to Mesh
Add creases to sharpen mesh edges.
You can add creases to mesh objects that have a smoothing level of 1 or higher.

Add Creases to Different Subobjects
The result of creasing differs, depending on what type of subobject you select.
- **Edge**. The selected edge is sharpened. The adjacent faces are deformed to accommodate the new crease angle.
- **Face**. The selected face is flattened and all edges that bound that face are sharpened. Adjacent faces are deformed to accommodate the new shape of the face.
- **Vertex**. The point of the vertex and all intersecting edges are sharpened. Adjacent faces are deformed to accommodate the new vertex angle.

Assign a Crease Value to the Edge
As you apply a crease, you set a crease value that determines how the crease is affected by smoothing. A value of Always ensures that the crease is always
retained, even when the mesh is repeatedly smoothed. Higher crease values ensure that the crease is retained through several smoothing processes. (During smoothing, the assigned crease value is decreased by the value of the original level of smoothing.)

You can add a crease to mesh that has not been smoothed. However, the effect is not visible unless you smooth the object.

**Remove a Crease**

You can restore a crease to a smoothed state that corresponds to the smoothing level for the object. If you remove a crease that is adjacent to other creased subobjects, their contours are adjusted.

**Quick Reference**

**Commands**

- **MESHCREASE**
  Sharpens the edges of selected mesh subobjects.
- **MESHUNCREASE**
  Removes the crease from selected mesh faces, edges, or vertices.

**PROPERTIES**

Controls properties of existing objects.

**Modify Mesh Faces**

Split, extrude, merge, collapse, or spin mesh faces to modify their shapes.

**Split a Mesh Face**

You can split a mesh face to make custom subdivisions. Use this method to prevent deforming a larger area for small modifications.
Because you specify the start point and end point of the split, this method also gives you control over the shape of the two new faces. Use the Vertex option to snap automatically to the vertices of the face. If you plan to split a face to create—and then spin the edge of—two triangular faces (MESHSPIN), use the Vertex option to ensure precision.

**Extrude Mesh Faces**

You can add definition to a 3D object by extruding a mesh face. Extruding other types of objects creates a separate 3D solid object. However extruding a mesh face extends, or deforms, the existing object and subdivides the extruded face.

You can use the same methods for extrusion of the faces of 3D solids and meshes as you use for other types of objects. For example, you can specify an extrusion direction, a path, or a taper angle. However, when you extrude mesh faces, the MESHEXTRUDE command provides an option that sets whether adjacent faces are extruded individually or whether their shared edges remained joined.
You cannot create joined extrusions for mesh faces in which only the vertices are shared.

For more information about extrusion, see Create a Solid or Surface by Extruding on page 450.

Reconfigure Adjacent Mesh Faces

You can extend your editing options by reconfiguring adjacent faces. Several options are available:

- **Merge adjacent faces.** Combine adjacent faces to form a single face. Merging works best with faces that are on the same plane.

  ![Merging of adjacent faces](image)

  Although you can merge faces that wrap a corner, additional modifications to the resulting mesh object can have unexpected results.

- **Collapse the mesh vertices.** Merge adjacent vertices of surrounding faces form a single point. The selected face is removed.

  ![Collapse of mesh vertices](image)
■ **Spin edges of triangular faces.** Rotate an edge that is shared by two triangular faces. The shared edge spins to extend from the opposite vertices. This activity works best when the adjoined triangles form a rectangular, not a triangular, shape.

![Diagram showing spin edges of triangular faces.]

See also:

■ **Tips for Working with Mesh** on page 627
■ **Create a Solid or Surface by Extruding** on page 450

**Quick Reference**

**Commands**

**MESHCOLLAPSE**

Merges the vertices of selected mesh faces or edges.

**MESHEXTRUDE**

Extends a mesh face into 3D space.

**MESHMERGE**

Merges adjacent faces into a single face.

**MESHSMOOTHLESS**

Decreases the level of smoothness for mesh objects by one level.

**MESHSMOOTHMORE**

Increases the level of smoothness for mesh objects by one level.

**MESHSPIN**

Spins the adjoining edge of two triangular mesh faces.
MESH_SPLIT
Splits a mesh face into two faces.

Create and Close Mesh Gaps
Delete mesh faces or close gaps in mesh objects.

Remove Mesh Faces
You can press Delete or use the ERASE command to remove mesh faces. The removal leaves a gap in the mesh.
- Deleting a face removes only the face.
- Deleting an edge removes each adjacent face.
- Deleting a vertex removes all faces that are shared by the vertex.
If removal of a mesh face creates a gap, the mesh object is not “watertight.” It can be converted to a surface object, but not to a 3D solid object.

**Close Gaps in Mesh Objects**

If a mesh object is not watertight due to gaps, or holes, in the mesh, you can make it watertight by closing the holes. The cap, or new face, spans the boundary formed by the mesh edges that you specify (MESHCAP).

This process works best when all edges are on the same plane. The edges you select as boundaries cannot be shared by two faces. For example, you cannot close the center hole in a mesh torus.

**NOTE** You can sometimes close gaps in mesh by smoothing the object, by using MESHCOLLAPSE, or by splitting adjacent faces (MESH_SPLIT).

**See also:**
- Tips for Working with Mesh on page 627

**Quick Reference**

**Commands**

ERASE

Removes objects from a drawing.

MESHCAP

Creates a mesh face that connects open edges.

**Tips for Working with Mesh**

Learn some best practices for working with mesh models.
Mesh, with its enhanced modeling capabilities, offers a way to create more fluid, free-form designs. Keep these tips in mind as you work.

**Model mesh before you smooth it.**

Mesh modeling is a powerful way to design, but higher levels of smoothness increase complexity and can affect performance. You can work more efficiently if you complete editing operations such as gizmo editing, extrusion, and face splitting, on mesh objects that have not been smoothed. (That is, their level of smoothness is 0.)

![mesh sphere modeled by grip editing and extrusion, then smoothed](image)

You can quickly switch between the levels of smoothness in the Properties Inspector palette to get a preview of how your activities affect the smoothed object.

**Refine or split a face instead of refining the entire object.**

Refinement is a powerful way to subdivide faces. However, by increasing the number of faces, you add to the overall complexity of the model. In addition, refining an entire mesh object resets the base level of smoothness to 0. This change can result in a dense grid that can no longer be simplified. For best results, avoid refining the object, and refine or split only the individual faces that require more detailed modeling.
Refining individual faces does not reset the level of smoothness for the object.

**Crease edges to help limit distortion when the object is smoothed.**

Creased edges can be set to maintain their sharpness, no matter how much the object is smoothed. You may also need to crease the edges in surrounding faces to obtain the result you want.

Creasing set to Always retains its sharpness after smoothing. If you set a crease value, the creased edge becomes smoother at the equivalent level of smoothness.

**Use gizmos to model faces, edges, and vertices.**

3D Move, 3D Rotate, and 3D Scale gizmos can be used to modify entire mesh objects, or specific subobjects.
For example, you can rotate and scale an individual face using the 3D Move, Rotate, and Scale gizmos.

By constraining the modifications to a specified axis or plane, gizmos help you avoid unexpected results. The default gizmo is displayed whenever you select an object in a view that uses a 3D visual style. (You can also suppress this display.) Therefore, you do not have to explicitly start the 3D Move, 3D Rotate, or 3D Scale command to initiate these activities. You just need to select an object.

When a gizmo is selected, you can use the shortcut menu to switch to a different type of gizmo.

*Use subobject selection filters to narrow the available selection candidates.*

In a smoothed mesh, trying to select a specific subject can be difficult unless you turn on subobject selection (shortcut menu). By specifying that the selection set is limited to faces, edges, vertices, or even solid history subobjects, you can restrict which subobject type is available for selection.

A filter is especially valuable for selecting mesh vertices, which are not highlighted as you move the mouse over them.
In order to select the entire mesh object, you need to turn off the subselection filters.

**Model by extruding faces.**

A key difference between gizmo editing and extrusion occurs in the way each face is modified. With gizmo editing, if you select and drag a set of faces, adjacent faces are stretched to accommodate the modification. When the object is smoothed, the adjacent faces adapt to the new location of the face.

Mesh extrusion, however, inserts additional faces to close the gap between the extruded face and its original surface. With mesh extrusion, you can set whether adjacent faces are extruded as a unit (joined) or separately (unjoined).

If you are working on an object that has not been smoothed, try smoothing it periodically to see how the extrusion is affected by smoothing.
Convert between mesh and 3D solids or surfaces.

Mesh modeling is powerful, but it cannot do everything that solid modeling can do. If you need to edit mesh objects through intersection, subtraction, or union, you can convert mesh to 3D solid or surface objects. Similarly, if you need to apply creasing or smoothing to 3D solid or surface objects, you can convert those objects to mesh.

Keep in mind that not all conversions retain complete fidelity to the shape of the original object. Avoid switching between object types more than once, if possible. If you notice that the conversion modifies the shape of the object in an unacceptable way, undo the conversion and try again with different settings.

The SMOOTHMESHCONVERT system variable sets whether the mesh objects that you convert to 3D solids or surfaces are smoothed or faceted, and whether their co-planar faces are optimized (merged).

You might have trouble converting some non-primitive mesh to solid objects due to the following problems:

- **Gaps in the mesh.** If you notice gaps, you can sometimes close them by smoothing the object or by refining the faces that are adjacent to the gap.

You can also close holes by using MESHCAP.

- **Intersecting mesh faces.** Be especially careful not to create self-intersections as you move, rotate, or scale subobjects. (You create self-intersections when you cause one or more faces to cross, or intersect other faces in the same mesh model.) View the object from all viewpoints to ensure you create a viable model.
Mesh objects that cannot be converted to solids can often be converted to surfaces instead.

**Avoid merging faces that wrap a corner**

When you merge faces, you can create a mesh configuration in which the merged face wraps a corner. If a resulting face has a vertex that has two edges and two faces, you cannot convert the mesh to a smooth 3D solid object.
One way to resolve this problem is to convert the mesh to a faceted solid instead of a smooth solid. You might also be able to repair the problem by splitting the adjacent faces, starting at the shared vertex (MESH_SPLIT).
Create Sections and 2D Drawings from 3D Models

Create cross sections, cutting planes, flattened views, and 2D drawings of 3D objects.

Work with Sections
Create cross sections of 3D models.

Overview of Section Objects
Create a section plane that can be modified and moved to achieve the cross section view that you need.

With the SECTIONPLANE command, you can create one or more section objects and place them throughout a 3D model (3D solids, surfaces, or mesh). By activating live sectioning, you can then view transient cuts in the 3D model as you move the section object through it. The 3D objects themselves do not change.

Set the Cross-Section with the Section Plane Indicator
Section objects have a transparent section plane indicator that acts as a cutting plane. This plane can be moved through a 3D model that is composed of 3D solids, surfaces, or regions to obtain different section views.
**Store Properties in Section Lines**

The section plane contains a *section line* that stores section object properties. You can create multiple section objects to store different properties. For example, one section object can display a hatch pattern at the section plane intersection. Another section object can display a specific linetype for the boundary of the intersected area.

**Analyze the Model with Live Sectioning**

With live sectioning, you can dynamically analyze the interior details of 3D objects by moving and adjusting the section plane. You can specify whether to hide, or cut away, the portion of the model that is on the viewing side of the section plane indicator.
Save and Share Section Images

After you create a sectional view, you can generate an accurate 2D or 3D block from the 3D model. These blocks can be analyzed or checked for clearances and interference conditions. They can also be dimensioned, or used as wireframe or rendered illustrations in documentation and presentation drawings.

Quick Reference

Commands

LIVESECTION

Turns on live sectioning for a selected section object.

SECTIONPLANE

Creates a section object that acts as a cutting plane through 3D objects.

SECTIONPLANESETTINGS

Sets display options for the selected section plane.

SECTIONPLANETOBlockly

Saves selected section planes as 2D or 3D blocks.

Create Section Objects

Create cross sections to show interior details of 3D objects.

With the SECTIONPLANE command, you create a section object that acts as a cutting plane through solids, surfaces, meshes, or regions. Then turn on live sectioning to move the section object through the 3D model to reveal its inner details in real time.

You can align a section object using several methods.

Align the Section Plane to a 3D Face

One way to set the section plane is to click the face of an existing 3D object. (As you move the cursor, a dotted outline indicates the side of the plane to be selected.) The section plane is automatically aligned to the plane of the face you select.
Create a Straight Cutting Plane

Pick two points to create a straight cutting plane.

Add a Jogged Segment

The section plane can be a straight line or it can have multiple or jogged sections. For example, a section containing a jog is one that cuts away a pie slice-shaped wedge from a cylinder.

Create a section line that has jogged segments by using the Draw Section option of SECTIONPLANE to pick multiple points throughout the 3D model.
Section object with jogged segment

Create Orthographic Sections
You can align section objects to a specified orthographic orientation of the current UCS, such as front, back, bottom, top, left, or right.

Orthographic section planes are placed so that they pass through the center of the 3D extents of all 3D objects in the drawing.

Create a Region to Represent the Cross Section
With the SECTION command, you can create a 2D region object that represents a planar cross section through a 3D solid object. You do not have live sectioning capabilities when you use this legacy method to create cross sections.
Define the plane of the cross section using one of the following methods:

- Specify three points
- Specify a 2D object such as a circle, ellipse, arc, spline, or polyline
- Specify a view
- Specify the Z axis
- Specify the XY, YZ, or ZX plane

The new region that represents the cross-sectional plane is placed on the current layer.

**NOTE** Before you apply hatching to the cross-sectional cutting plane, align the UCS with the cutting plane.

---

**Quick Reference**

**Commands**

**SECTIONPLANE**

Creates a section object that acts as a cutting plane through 3D objects.

**SECTION**

Uses the intersection of a plane and solids, surfaces, or mesh to create a region.

---

**Modify a Section View**

After you create a section, adjust its display or modify its shape and location to change the represented section view.
Add Jogs to a Section

Add jogs, or angular segments, to existing section lines.

You can create a section plane that has multiple segments (jogs), using the Draw Section option of the SECTIONPLANE command. You can also add a jog to an existing section plane by selecting the section you want to add a job to and then right-clicking. From the shortcut menu, click Add Jog to Section (SECTIONPLANEJOG).

A jog that is added to an existing section object creates a segment that is perpendicular to the selected segment. Its viewpoint is oriented in the direction set by the Direction grip. The Nearest object snap is temporarily turned on to help you place the jogs on a section.

You cannot add jogs to the side or back lines of the section object.

After adding jogs, you can reposition and resize the jogged sections by dragging the section object grips.

Quick Reference

Commands

SECTIONPLANE

Create a section object that acts as a cutting plane through 3D objects.

SECTIONPLANEJOG

Adds a jogged segment to a section object.

Use Live Section to Adjust the Cross Section

Use live sectioning to move a section object through the 3D model or region dynamically.
What Is Live Sectioning?

Live sectioning is an analytical tool for viewing cut geometry in a 3D solid, surface, or region.

You can use live sectioning to analyze a model by moving the section object through the object. For example, sliding the section object through an engine assembly helps you visualize its internal components. You can use this method to create a cross section view that you can save or reuse.

Turn on and Use Live Sectioning

Live sectioning works with 3D objects and regions in model space. When live sectioning is activated, you can change the viewing planes by using grips to adjust the location of the section object or its segments.

By turning on cutaway geometry, you can display the entire object that contains the section plane. This option (available on the shortcut menu) can only be turned on when section plane is active.

Live sectioning is turned on or off automatically, depending on how you create the section object. For example, when you select a face to define the section plane, live sectioning is turned on. When you create sections using
the Draw Section option of the SECTIONPLANE command, live sectioning is turned off. Live sectioning can be manually turned on or off after a section object is created.

A drawing can contain multiple section objects. However, live sectioning can only be active for one section object at a time. Suppose that your model has two sections objects: Section A and Section B. If Section A has live sectioning turned on and you activate live sectioning for Section B, live sectioning for Section A is automatically turned off.

Turning off a section object layer does not turn off live sectioning. However, freezing the layer turns off live sectioning.

**Quick Reference**

**Commands**

LIVESECTION

Turns on live sectioning for a selected section object.

SECTIONPLANE

Creates a section object that acts as a cutting plane through 3D objects.

SECTIONPLANESETTINGS

Sets display options for the selected section plane.

**Use Grips to Modify Section Objects**

Section object grips help you move and resize the section object.
Grips allow you to adjust the location, length, width, and height of the cutting area.

- **Base grip.** Acts as the basepoint for moving, scaling, and rotating the section object. It is always adjacent to the Menu grip.

- **Second grip.** Rotates the section object around the base grip.

- **Menu grip.** Displays a menu of section object states, which control the display of visual information about the cutting plane.

- **Direction grip.** Controls the viewing direction of the 2D section. To reverse the viewing direction of the section plane, click the Direction grip.

- **Arrow grip.** Modifies the section object by modifying the shape and position of the section plane. Only orthogonal movements in the direction of the arrow are permitted. (Section Boundary and Volume states only.)

- **Segment end grips.** Stretches the vertices of the section plane. You cannot move segment end grips so that segments intersect. Segment end grips are displayed at the endpoints of jogged segments. (Section Boundary and Volume states only.)

You can select only one section object grip at a time.
Quick Reference

Commands
SECTIONPLANE
Creates a section object that acts as a cutting plane through 3D objects.

System Variables
GRIPSIZE
Sets the size of the grip box in pixels.

Set Section Object States and Properties
Set the display of the section object.

Set Section Object States

Section objects have the following display states:

- **Section Plane.** The section line and transparent section plane indicator are displayed. The cutting plane extends infinitely in all directions.

- **Section Boundary.** A 2D box shows the XY extents of the cutting plane. The cutting plane along the Z axis extends infinitely.

- **Section Volume.** A 3D box shows the extents of the cutting plane in all directions.

You can switch between object states by clicking the Menu grip that is displayed when you select the section object.
Set Section Object Properties

Section objects have properties like other AutoCAD objects. Properties are stored in the section line and can be accessed in the Properties Inspector.

For each section object, you can change the name, layer, and linetype. You can also change the color and transparency of the section plane indicator (the cutting plane).

Quick Reference

Commands

PROPERTIES
Controls properties of existing objects.

SECTIONPLANE
Creates a section object that acts as a cutting plane through 3D objects.

SECTIONPLANESETTINGS
Sets display options for the selected section plane.

SECTIONPLANETOBLOCK
Saves selected section planes as 2D or 3D blocks.

Associate Section Objects with Named Views

Associate section objects with named views.

When you activate a named view that has an associated section object, live sectioning is turned on for that section object. For a 3D model with multiple section objects, you might want to associate a particular section object to a view. Later, you can restore a saved sectional view and activate live sectioning for the associated section object.

For example, you can set up two section objects that cut through the 3D model in different directions. Section object A cuts the model along its width; Section object B cuts the model along its length. Perhaps you want to view the sectional cut that is perpendicular to your line of sight. By associating each section object with a view, you can quickly switch between the two views and see the desired cross section.
Quick Reference

Commands

SECTIONPLANE

Creates a section object that acts as a cutting plane through 3D objects.

VIEW

Save and Publish Section Objects

Save a section object as a block or publish it.

Save Sections as Blocks or Drawings

Save the representation of the cross-sectional area where a section object intersects a 3D model as a block.

Save Sections as Blocks or Drawings

You can save the section objects you create as blocks. Working from the Generate Section/Elevation dialog box, you can choose the type of block that is created.
For example, suppose your project requires 2D elevation drawings or 2D cross sections. The 2D Section / Elevation option creates an accurate block representation that is ready for dimensioning.

To publish or render a cutaway of the 3D model, select the 3D Section option. 3D section geometry consists of mostly 3D solids and surfaces. However, profile outlines and hatch patterns consist of 2D lines.

The display properties of 2D section/elevation blocks and 3D section blocks are controlled in the Section Settings dialog box.

When you create section blocks, you have the following choices for how they are handled:

- **Insert the section blocks.** At the time of creation, you can insert a 2D or 3D section block into the drawing or save it to an external file. A 2D section block is inserted on the XY plane of the current UCS, including section blocks that extend into 3D space. Inserted section blocks are initially unnamed. You can set the scale, rotation, and basepoint upon insertion. You can modify and rename them later by editing the block with BEDIT.

- **Export section blocks to a file.** Save and name the new section objects so they can be inserted later.
■ **Save section block components on separate layers.** By default, section block components such as intersection boundary, intersection fill, background lines, cutaway geometry, and curve tangency lines are saved on Layer 0. However, you can separate the components of saved section blocks onto separate layers with a suffix or prefix that you specify. Assigning a suffix or prefix helps you organize the block components into layers that you can sort and identify quickly. The Layer properties lists in the Section Settings dialog box provide the opportunity to customize the layer names.

■ **Specify whether to limit the section block to certain objects.** The objects that are included in a section block vary, depending on which section object state is selected. You can also select specific objects to be included as you create the section block.

**Quick Reference**

**Commands**

**BEDIT**

Opens the block definition in the Block Editor.

**SECTIONPLANE**

Creates a section object that acts as a cutting plane through 3D objects.

**SECTIONPLANESETTINGS**

Sets display options for the selected section plane.

**SECTIONPLANETOBLOCK**

Saves selected section planes as 2D or 3D blocks.

**Publish Section Objects**

Control the visibility of section objects when you render or print.

**Render Section Objects**

With live sectioning turned on, all lines on a section object are rendered as 2D lines. The section plane indicator is rendered as a transparent material. Its degree of transparency is controlled in the Properties Inspector.
If you want to render a 3D cutaway, save the cutaway section as a 3D block and render the block reference.

Print Section Objects
When a section object is in a Section Boundary or Section Volume state, displayed lines cannot be printed. The section plane indicator is printed as if it were transparent. However, it does not have the same visual quality that it has when it is rendered.
If you do not want to plot the section line, place the section object on a layer that is turned off.

Quick Reference

Commands
PLOT
Outputs a drawing to a printer or file.
RENDER
Creates a photorealistic or realistically shaded image of a 3D solid or surface model.
SECTIONPLANE
Creates a section object that acts as a cutting plane through 3D objects.
SECTIONPLANESETTINGS
Sets display options for the selected section plane.
SECTIONPLANETO.block
Saves selected section planes as 2D or 3D blocks.

Create a Flattened View
Create a flattened view of the 3D solids and regions in the current view.

Create a 2D Presentation of a 3D Model
With the FLATSHOT command, you can create a flattened, 2D representation of the 3D model projected onto the XY plane. The resulting objects can be inserted as a block or saved as a separate drawing.
The process is like taking a photograph of the entire 3D model and then laying the photograph flat. This feature is useful for creating technical illustrations.

The flatshot process works only in model space. Start by setting up the view you want, including orthographic or parallel views. All 3D objects in the model space viewport are captured. Therefore, be sure to place the objects you do not want captured on layers that are turned off or frozen.

As you create the block, you can control how hidden lines are displayed by adjusting the Foreground and Obscured Lines settings in the Flatshot dialog box. For best results with mesh objects, clear the Show box under Obscured Lines so that hidden lines are not represented.

Three-dimensional objects that have been sectioned are captured in their entirety, as if they had not been sectioned.

**NOTE** To create profile images of 3D solids in paper space, use the SOLPROF command.

**Modify a Block Created with Flatshot**

You can modify a flattened view that has been inserted as a block in the same way that you modify any other 2D block geometry.
Quick Reference

Commands

FLATSHOT
  Creates a 2D representation of all 3D objects based on the current view.

SOLPROF
  Creates 2D profile images of 3D solids for display in a layout viewport.

SOLDRAW
  Generates profiles and sections in layout viewports created with SOLVIEW.

SOLVIEW
  Creates orthographic views, layers, and layout viewports automatically for
  3D solids.
Annotate Drawings
Work with Annotations

When you annotate your drawings, you can use certain tools and properties to make working with annotations easier.

Overview of Annotations

Annotations are notes or other types of explanatory symbols or objects that are commonly used to add information to your drawing.

Examples of annotations include

- Notes and labels
- Tables
- Dimensions and tolerances
- Hatches
- Callouts
- Blocks

The types of objects that you use to create annotations include

- Hatches
- Text (single-line and multiline)
- Tables
- Dimensions
- Tolerances
- Leaders and multileaders
Blocks
Attributes

Quick Reference

Commands

ATTDEF
Creates an attribute definition for storing data in a block.

BLOCK
Creates a block definition from selected objects.

DIMSTYLE
Creates and modifies dimension styles.

HATCH
Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

MLEADERSTYLE
Creates and modifies multileader styles.

MTEXT
Creates a multilined text object.

OBJECTSANSLE
Adds or deletes supported scales for annotative objects.

STYLE
Creates, modifies, or specifies text styles.

TEXT
Creates a single-line text object.

Scale Annotations
You can automate the process of scaling annotations in various layout viewports and in model space.
Overview of Scaling Annotations

Objects that are commonly used to annotate drawings have a property called . This property allows you to automate the process of scaling annotations so that they plot or display at the correct size on the paper.

Instead of creating multiple annotations at different sizes and on separate layers, you can turn on the annotative property by object or by style, and set the annotation scale for model or layout viewports. The annotation scale controls the size of the annotative objects relative to the model geometry in the drawing.

The following objects are commonly used to annotate drawings and contain an annotative property:

- Text
- Dimensions
- Hatches
- Tolerances
- Multileaders
- Blocks
- Attributes

When the Annotative property for these objects is turned on (set to Yes), these objects are called annotative objects.

You define a paper size for annotative objects. The you set for layout viewports and model space determines the size of the annotative objects in those spaces.

Save to Legacy Drawing File Format

Set the system variable SAVEFIDELITY to 1 when you save a drawing that contains annotative objects to a legacy drawing file format (AutoCAD 2007 or earlier). This preserves the visual fidelity of the drawing when it is opened in a release earlier than AutoCAD 2008 by saving individual representations of each scale of each annotative object. The individual objects are saved to layers that are used to organize objects of the same scale. Setting SAVEFIDELITY to 0, when opening the drawing in AutoCAD 2008 or later release, results in improved performance. For more information about saving a drawing to a previous release, see Save a Drawing on page 67.
Quick Reference

Commands
OBJECTSCALE
Adds or deletes supported scales for annotative objects.

System Variables
ANNOAUTOSCALE
Updates annotative objects to support the annotation scale when the annotation scale is changed.
CANNOSCALE
Sets the name of the current annotation scale for the current space.
CANNOSCALEVALUE
Returns the value of the current annotation scale.
MSLTSCALE
Scales linetypes displayed on the model tab by the annotation scale.

Set Annotation Scale
is a setting that is saved with model space, layout viewports, and model views. When you add objects to your drawing, they support the current annotation scale and are scaled based on that scale setting and automatically displayed at the correct size in model space.

Before you add annotative objects to your model, you set the annotation scale. Think about the eventual scale settings of the viewports in which the annotations will display. The annotation scale should be set to the same scale as the viewport in which the annotative objects will display in the layout (or the print scale if printing from model space). For example, if the annotative objects will display in a viewport that has a scale of 1:2, then you set the annotation scale to 1:2.

When working on the Model layout or when a viewport is selected, the current annotation scale is displayed on the status bar. You can use the status bars to change the annotation scale. You can reset the annotation scale list to the default list of scales stored with your user profile in the Default Scale List dialog box.
You can use the ANNOAUTOSCALE system variable to update annotative objects to support the current scale automatically when the annotation scale is changed. ANNOAUTOSCALE is turned off by default to keep file size down and improve performance. When ANNOAUTOSCALE is off, this button is displayed this way on the right side of the status bar.

Use the CANNOSCALE system variable to set a default annotation scale setting.

You can reset the list of annotative scales in a drawing to the default list of either metric or imperial scales defined in the registry with the Default Scale dialog box. The unused scales in the drawing are purged and the customized list of scales from your user profile are merged into the drawing.

See also:

- The Status Bar on page 35

Quick Reference

Commands

OBJECTSCALE

Adds or deletes supported scales for annotative objects.

System Variables

ANNOAUTOSCALE

Updates annotative objects to support the annotation scale when the annotation scale is changed.

CANNOSCALE

Sets the name of the current annotation scale for the current space.

CANNOSCALEVALUE

Returns the value of the current annotation scale.

MSLTSCALE

Scales linetypes displayed on the model tab by the annotation scale.
Create Annotative Objects

Objects that are commonly used to annotate drawings have a property called . When the Annotative property for these objects is turned on (set to Yes), these objects are called annotative objects

Overview of Creating Annotative Objects

When you add annotations to your drawing, you can turn on the property for those objects. These annotative objects are scaled based on the current setting and are automatically displayed at the correct size

Annotative objects are defined at a paper height and display at the size determined by the annotation scale.

The following objects can be annotative (have an Annotative property):

- Hatches
- Text (single-line and multiline)
- Dimensions
- Tolerances
- Leaders and multileaders (created with MLEADER)
- Blocks
- Attributes

Many of the dialog boxes used to create these objects contain an Annotative check box where you can make the object annotative. You can also change existing objects to be annotative by changing the annotative property in the Properties Inspector palette.

When you hover the cursor over an annotative object that supports one annotation scale, the cursor displays a icon. When the object supports more than one annotation scale, it displays a icon.

Text, dimension, and multileader styles can also be annotative. Annotative styles create annotative objects.
Visual Fidelity for Annotative Objects

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

If you work primarily in model space, it is recommended that you turn off visual fidelity (set SAVEFIDELITY to 0). However, if you need to exchange drawings with other users, and layout fidelity is most important, then visual fidelity should be turned on (set SAVEFIDELITY to 1).

NOTE The SAVEFIDELITY system variable does not affect saving a drawing to the AutoCAD 2010 drawing or DXF file formats.

Annotative objects may have multiple . When visual fidelity is on, annotative objects are decomposed and scale representations are saved (in an ) to separate layers, which are named based on their original layer and appended with a number. If you explode the block in AutoCAD 2007 or earlier releases, and then open the drawing in AutoCAD 2008 or later releases, each scale representation becomes a separate annotative object, each with one annotation scale. It is not recommended that you edit or create objects on these layers when working with a drawing created in AutoCAD 2008 and later releases in AutoCAD 2007 and earlier releases.

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

See also:

- Work with Annotative Styles on page 662

Quick Reference

Commands

ATTDEF

Creates an attribute definition for storing data in a block.

BLOCK

Creates a block definition from selected objects.
DIMSTYLE
Creates and modifies dimension styles.

HATCH
Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

MLEADERSTYLE
Creates and modifies multileader styles.

MTEXT
Creates a multiline text object.

STYLE
Creates, modifies, or specifies text styles.

TEXT
Creates a single-line text object.

Work with Annotative Styles
You can minimize the steps to annotate a drawing by using annotative styles.
Annotative text, dimension, and multileader styles create objects.
The dialog boxes used to define these objects contain an Annotative check box where you can make the styles annotative. Annotative styles display a special icon before their names in dialog boxes and the Properties Inspector palette.

You should specify the Paper Text Height value for any annotative text styles you create. The Paper Text Height setting specifies the height of the text in paper space.

NOTE If you’ve specified the Text Height value for a dimension or multileader style, this setting overrides the text style Paper Text Height setting.

If you redefine styles to be annotative or non-annotative, existing objects that reference those styles are not automatically updated to reflect the annotative property of the style or definition. Use the ANNOUPDATE command to update the existing objects to the current annotative properties of the style.
When you change the Style property of an existing object (whether it's annotative or non-annotative), the object's annotative properties will match that of the new style. If the style does not have a fixed height (the Text Height value is 0), the paper height of the object is calculated based on the object's current height and the annotation scale.

See also:

- Work with Text Styles on page 738
- Create Annotative Text on page 663
- Use Dimension Styles on page 773
- Create Annotative Dimensions and Tolerances on page 664
- Work with Leader Styles on page 728
- Create Annotative Leaders and Multileaders on page 666

Quick Reference

Commands

ANNOUPDATE

Updates existing annotative objects to match the current properties of their styles.

DIMSTYLE

Creates and modifies dimension styles.

MLEADERSTYLE

Creates and modifies multileader styles.

STYLE

Creates, modifies, or specifies text styles.

Create Annotative Text

Use text for notes and labels in your drawing. You create annotative text by using an annotative text style, which sets the height of the text on the paper.
The current automatically determines the display size of the text in model space or paper space viewports.

For example, you want text to display at a height of 3/16" on the paper, so you can define a text style to have a paper height of 3/16". When you add text to a viewport that has a scale of 1/2"=1'0", the current annotation scale, which is set to the same scale as the viewport’s, automatically scales the text to display appropriately at 4.5".

You can also change existing non-annotative text to annotative by changing the text’s Annotative property to Yes (On). This applies to any text created through text styles or through the TEXT and MTEXT commands.

You can set the orientation of annotative text objects to match the orientation of the paper. For more information about setting the orientation of annotative objects, see Set Orientation for Annotations on page 673.

See also:
- Create Text on page 706
- Work with Annotative Styles on page 662
- Set Orientation for Annotations on page 673

Quick Reference

Commands

MTEXT
  Creates a multiline text object.

STYLE
  Creates, modifies, or specifies text styles.

TEXT
  Creates a single-line text object.

Create Annotative Dimensions and Tolerances

You can create dimensions for measurements in your drawing through annotative dimension styles.
Annotative dimension styles create dimensions in which all the elements of the dimension, such as text, spacing, and arrows, scale uniformly by the .

If you associate a dimension to an annotative object, the associativity of the dimension is lost.

You can also change an existing non-annotative dimension to annotative by changing the dimension’s Annotative property to Yes (On).

**NOTE** When the current dimension style is annotative, the value of DIMSCALE is automatically set to zero, and does not affect the dimension scale.

You can also create annotative tolerances. Geometric tolerances show acceptable deviations of form, profile, orientation, location, and runout of a feature.

**See also:**
- Dimensions and Tolerances on page 767
- Use Dimension Styles on page 773
- Work with Annotative Styles on page 662

**Quick Reference**

**Commands**

DIMSTYLE

Creates and modifies dimension styles.
STYLE

Creates, modifies, or specifies text styles.

System Variables

DIMANNO

Creates a single-line text object.

Create Annotative Leaders and Multileaders

Leaders and multileader on page 1023 are used to add call outs to your drawings. You can create leaders through an annotative dimension style and multileaders through an annotative multileader style.

When you create a leader, you create two separate objects: the leader and the text, block, or tolerance associated with the leader. When you create a multileader, you create a single object.

If the multileader style is annotative, the associated text or tolerance will be annotative as well, regardless of the annotative setting of the text style or tolerance.

NOTE It is recommended that you create non-annotative entities when creating a mleader content block.

Blocks used in leaders and multileaders must be non-annotative.

You can change the Annotative property of leaders and multileaders in the Properties Inspector palette.

See also:

■ Create Leaders on page 723
■ Work with Leader Styles on page 728

Quick Reference

Commands

MLEADERSTYLE

Creates and modifies multileader styles.
STYLE

Creates, modifies, or specifies text styles.

Create Annotative Blocks and Attributes

If you want to use geometric objects to annotate your drawing, combine the objects into an annotative block definition.

Block definitions create annotative block references. Annotative block references and attributes initially support the current annotation scale at the time they are inserted. You should insert annotative block references with a unit factor of 1.

You cannot change the Annotative property of individual block references.

To set an annotative block’s paper size, you should define the block in paper space or on the Model layout with the set to 1:1.

When creating and working with annotative blocks and annotative objects within blocks, the following points should be noted:

- Non-annotative blocks can contain annotative objects, which are scaled by the block’s scale factor in addition to the annotation scale.
- Annotative blocks cannot reside in annotative blocks.
Annotative block references are scaled uniformly by the current annotation scale as well as any user scale applied to the block reference.

Blocks that contain annotative objects should not be manually scaled.

You can define annotative attributes for annotative and non-annotative blocks. Use annotative attributes with non-annotative blocks when you want the geometry in the block to display on the paper based on the scale of the viewport, but you want the attribute text to display at the Paper Text Height defined for the attribute.

You can set the orientation of annotative blocks to match the orientation of the paper. For more information about setting the orientation of annotative objects, see Set Orientation for Annotations on page 673.

You can use the ANNOTATIVEDWG system variable to specify whether or not the entire drawing will behave as an annotative block when inserted into another drawing. The ANNOTATIVEDWG system variable becomes read-only if the drawing contains annotative objects.

NOTE The INSUNITS setting is ignored when inserting blocks into a drawing.

See also:

- Work with Blocks on page 405
- Attach Data to Blocks (Block Attributes) on page 420
- Set Orientation for Annotations on page 673

Quick Reference

Commands

ATTDEF
Creates an attribute definition for storing data in a block.

BLOCK
Creates a block definition from selected objects.
**System Variables**

ANNOTATIVEDWG

Specifies whether or not the drawing will behave as an annotative block when inserted into another drawing.

**Create Annotative Hatches**

Use an annotative hatch to symbolically represent material such as sand, concrete, steel, earth, etc.

An hatch is defined at a paper size. You can create individual annotative hatch objects as well as annotative hatch patterns.

The hatch pattern definitions stored in the acad.pat file contain information that indicates whether the pattern is annotative or non-annotative. When the selected hatch pattern is annotative, the Annotative checkbox in the Hatch and Gradient dialog box should be selected.

You can use the HPANNOTATIVE system variable to specify whether or not new hatch objects are annotative. By default, new hatch objects are non-annotative.

The orientation of annotative hatches always matches the orientation of the layout.
See also:

- Overview of Hatch Pattern Definitions in the Customization Guide

Quick Reference

Commands

HATCH

Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

System Variables

HPANNOTATIVE

Display Annotative Objects

For model space or a layout viewport, you can display all the annotative objects or only those that support the current annotation scale.

This reduces the need to use multiple layers to manage the visibility of your annotations.

You use the Annotation Visibility button on the right side of the application or drawing status bar to choose the display setting for annotative objects.

Annotation visibility is turned on by default. When annotation visibility is turned on, all annotative objects are displayed. When annotation visibility is turned off, only annotative objects for the current scale are displayed.

In general, you should turn off annotation visibility, except when inspecting a drawing created by another person or when adding scales to existing annotative objects.

Annotation visibility is also controlled by the ANNOALLVISIBLE system variable.

In order for an annotative object to be visible, the layer the object is on must be turned on.
If an object supports more than one annotation scale, the object will display at the current scale.

When the MSLTSCALE system variable is set to 1 (default), linetypes displayed on the model tab are scaled by the annotation scale.

See also:
- The Application Menu

**Quick Reference**

**System Variables**

ANNOALLVISIBLE
- Hides or displays annotative objects that do not support the current annotation scale.

MSLTSCALE
- Scales linetypes displayed on the model tab by the annotation scale.

SELECTIONANNODISPLAY
- Controls whether alternate scale representations are temporarily displayed in a dimmed state when an annotative object is selected.

**Add and Modify Scale Representations**

When you create an object in your drawing, it supports one: the annotation scale that was current when you created the object. You can update annotative objects to support additional annotation scales.

When you update an annotative object to support additional scales, you add additional to the object.

For example, if an annotative multileader supports two annotation scales, it has two scale representations.

When you select an annotative object, grips are displayed on the scale representation that supports the current annotation scale. You can use these grips to manipulate the current scale representation. All other scale representations of the object are displayed in a dimmed state when the SELECTIONANNODISPLAY system variable is set to 1 (default).
Use the ANNORESET command to reset the location of all scale representations for an annotative object to that of the current scale representation.

**Quick Reference**

**Commands**

- **ANNORESET**
  
  Resets the locations of all alternate scale representations of the selected annotative objects.

- **OBJECTSCALE**
  
  Adds or deletes supported scales for annotative objects.

**System Variables**

- **SELECTIONANNODISPLAY**
  
  Controls whether alternate scale representations are temporarily displayed in a dimmed state when an annotative object is selected.
Set Orientation for Annotations

blocks and text can be set so that their orientation matches the orientation of the layout. The orientation of annotative hatches always matches the orientation of the layout.

Even if the view in the layout viewport is twisted or if the viewpoint is non-planar, the orientation of these objects in layout viewports will match the orientation of the layout.

Annotative attributes in blocks match the paper orientation of the block.

See also:

- Work with Text Styles on page 738
- Create Annotative Text on page 663
- Create Annotative Blocks and Attributes on page 667
- Create Annotative Hatches on page 669
Hatches, Fills, and Wipeouts

- Overview of Hatch Patterns and Fills on page 675
- Specify Hatch and Fill Areas on page 680
- Control the Appearance of Hatches on page 684
- Modify Hatches and Fills on page 697
- Create a Blank Area to Cover Objects on page 702

Overview of Hatch Patterns and Fills

Hatches and fills do not have to be bounded. In the following illustration, the concrete hatches are *bounded*, while the earth hatches are *unbounded*.
By default, bounded hatches are *associative*, which means that the hatch object is associated with the hatch boundary objects, and changes to the boundary objects are automatically applied to the hatch.

To maintain associativity, the boundary objects must continue to completely enclose the hatch.

The alignment and orientation of a hatch pattern is determined by the current location and orientation of the user coordinate system, in addition to controls in the user interface.

Moving or rotating the UCS is an alternate method for controlling hatch patterns.

**NOTE** By default, a preview of the hatch displays as you move the cursor over enclosed areas. To improve the response time in large drawings, turn off the hatch preview feature with the HPQUICKPREVIEW system variable, or decrease the time before the preview is temporarily canceled with the HPQUICKPREVTIMEOUT system variable.

Alternatively, solid-filled areas can be created using

- 2D solids (SOLID)
- Wide polylines (PLINE)
- Donuts (DONUT)
- Traces (TRACE)
Quick Reference

Commands

GRADIENT
Fills an enclosed area or selected objects with a gradient fill.

HATCH
Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

HATCHEDIT
Modifies an existing hatch or fill.

HATCHTOBACK
Sets the draw order for all hatches in the drawing to be behind all other objects.

MATCHPROP
Applies the properties of a selected object to other objects.

PROPERTIES
Controls properties of existing objects.

UCS
Manages user coordinate systems.

System Variables

GFANG
Specifies the angle of a gradient fill.

GFCLR1
Specifies the color for a one-color gradient fill or the first color for a two-color gradient fill.

GFCLR2
Specifies the second color for a two-color gradient fill.

GFCLRLUM
Controls the tint or shade level in a one-color gradient fill.
GFCLRSTATE
  Specifies whether a gradient fill uses one color or two colors.
GFNAME
  Specifies the pattern of a gradient fill.
GFSHIFT
  Specifies whether the pattern in a gradient fill is centered or is shifted up
  and to the left.
HPANG
  Sets the angle for new hatch patterns.
HPANNOTATIVE
  Controls whether a new hatch pattern is annotative.
HPASSOC
  Controls whether hatches and fills are associative.
HPBACKGROUNDCOLOR
  Controls the background color for hatch patterns.
HPBOUND
  Controls the object type created by HATCH and BOUNDARY.
HPBOUNDRETAIN
  Controls whether boundary objects are created for new hatches and fills.
HPCOLOR
  Sets a default color for new hatches.
HPDOUBLE
  Specifies hatch pattern doubling for user-defined patterns.
HPDRAWORDER
  Controls the draw order of hatches and fills.
HPGAPTOL
  Treats a set of objects that almost enclose an area as a closed hatch boundary.
HPINHERIT
Controls whether to inherit the hatch origin when using the Inherit Properties option in HATCH and HATCHEDIT.

HPISLANDDETECTION
Controls how islands within the hatch boundary are treated.

HPISLANDDETECTIONMODE
Controls whether internal closed boundaries, called islands, are detected.

HPLAYER
Specifies a default layer for new hatches and fills.

HPMAXLINES
Sets the maximum number of hatch lines that are generated in a hatch operation.

HPNAME
Sets the default hatch pattern name.

HPOBJWARNING
Sets the number of hatch boundary objects that can be selected before displaying a warning message.

HPORIGIN
Sets the hatch origin point for new hatch patterns relative to the current user coordinate system.

HPORIGINMODE
Controls how the default hatch origin point is determined.

HPQUICKPREVIEW
Controls whether to display a preview when specifying internal points for a hatch.

HPQUICKPREVTIMEOUT
Sets the maximum duration that the hatch preview should be displayed before the preview is canceled.

HPSCALE
Sets the hatch pattern scale factor.
HPSEPARATE
Controls whether a single hatch object or separate hatch objects are created when operating on several closed boundaries.

HPSPACE
Sets the hatch pattern line spacing for user-defined patterns.

HPTRANSPARENCY
Sets the default transparency for new hatches and fills.

MIRRHATCH
Controls how MIRROR reflects hatch patterns.

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

Specify Hatch and Fill Areas
Define boundaries for hatches and fills from existing objects or from specified boundary points.
Use one of several methods to specify the 2D geometric boundaries of a hatch or fill.

■ Specify a point in an area that is enclosed by objects.
■ Select objects that enclose an area.
■ Specify boundary points using the Draw option of the -HATCH command.

NOTE Enclosed areas can be hatched only if they are in a plane parallel to the XY plane of the current UCS.

Create Associative Hatches
Associative hatches are automatically updated when their boundary objects are modified. Minor changes in the boundary of an associative hatch do not require erasing and re-creating the hatch.
Hatch associativity is turned on by default and is controlled by the HPASSOC system variable. You can also control hatch associativity using the following tools in the user interface:

- Hatch and Gradient dialog box
- Hatch Edit dialog box
- Properties Inspector

Nonassociative hatches are not updated when their original boundary is changed.

**Hatch Enclosed Areas Within Boundaries**

Enclosed areas within hatch boundaries are called *islands*. There are four island detection styles available from the user interface:

- Normal Island Detection
- Outer Island Detection (recommended)
- Ignore Island Detection
- No Island Detection (legacy behavior that is similar to the Ignore style)

Using Normal Island Detection, if you specify the internal pick point shown, islands remain unhatched and islands within islands are hatched.
Using the same pick point, the results of the options are compared below.

**NOTE** Text objects are treated as islands. If island detection is turned on, the result always leaves a rectangular space around the text.

**Include Objects in a Boundary Set**
When hatching a small area in a large, complex drawing, you can save time by selecting a smaller set of objects in the drawing to be used in determining the hatch boundary.

**Identify Gaps in Hatch Boundaries**
If the specified internal point is not within a fully enclosed area, red circles are displayed at the unconnected endpoints of the boundary to identify the gaps.
The red circles remain displayed after you exit the HATCH command. They are removed when you specify another internal point for the hatch, or when you use the REDRAW, REGEN, or REGENALL commands.

To hatch an area whose boundary is not quite closed, do one of the following:

- Locate the gaps and modify the boundary objects so they form a closed boundary.
- Set the HPGAPTOL system variable to a value large enough to bridge the gaps. HPGAPTOL applies only to gaps between geometric objects that would meet if extended.

**NOTE** By default, a preview of the hatch displays as you move the cursor over bounded areas. To improve the response time in large drawings, turn off the hatch preview feature with the HPQUICKPREVIEW system variable, or decrease the time before the preview is temporarily canceled with the HPQUICKPREVTIMEOUT system variable.

See also:

- Reshape a Hatch or Fill on page 699

**Quick Reference**

**Commands**

**HATCH**

Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

**HATCHEDIT**

Modifies an existing hatch or fill.
**System Variables**

HPANG
Sets the angle for new hatch patterns.

HPASSOC
Controls whether hatches and fills are associative.

HPBOUND
Controls the object type created by HATCH and BOUNDARY.

HPBOUNDRETAIN
Controls whether boundary objects are created for new hatches and fills.

HPGAPTOL
Treats a set of objects that almost enclose an area as a closed hatch boundary.

HPISLANDDETECTION
Controls how islands within the hatch boundary are treated.

HPISLANDDETECTIONMODE
Controls whether internal closed boundaries, called islands, are detected.

HPQUICKPREVIEW
Controls whether to display a preview when specifying internal points for a hatch.

HPQUICKPREVTIMEOUT
Sets the maximum duration that the hatch preview should be displayed before the preview is canceled.

HPSEPARATE
Controls whether a single hatch object or separate hatch objects are created when operating on several closed boundaries.

---

**Control the Appearance of Hatches**

Specify a hatch pattern or fill, and control its alignment and scale.
**Choose a Hatch Pattern or Fill**

Choose from three types of hatch patterns, and two types of fills.

- **Predefined hatch patterns.** Choose from over 70 ANSI, ISO, and other industry-standard hatch patterns that are available. You can also use hatch patterns from hatch pattern libraries supplied by other companies. Hatch patterns are defined in the `acad.pat` and `acadiso.pat` files.

- **User-defined hatch patterns.** Define a hatch pattern that uses the current linetype with a specified spacing and angle.

- **Custom hatch patterns.** Define a custom hatch pattern definition in a `.pat` file.

- **Solid fill.** Fill an area with a solid color by choosing the SOLID predefined hatch.

- **Gradient fill.** Fill an enclosed area with a color gradient. A gradient fill can be displayed as a *tint* (a color mixed with white), a *shade* (a color mixed with black), or a smooth transition between two colors.

Gradients that mimic colors displayed on a cylinder, a sphere, or other shapes are available.

**NOTE** You cannot use plot styles to control the printed color of gradient fills.

**Assign a Background Color to Hatch Patterns**

Predefined, user defined, and custom hatch patterns, can be assigned a background fill color. The background fill color shares the same level of transparency as the pattern itself.

**See also:**

- **Modify Hatch Properties** on page 697

- “Overview of Hatch Pattern Definitions” in the *Customization Guide*
Quick Reference

Commands

GRADIENT
Fills an enclosed area or selected objects with a gradient fill.

HATCH
Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

HATCHEDIT
Modifies an existing hatch or fill.

HATCHTOBACK
Sets the draw order for all hatches in the drawing to be behind all other objects.

MATCHPROP
Applies the properties of a selected object to other objects.

PROPERTIES
Controls properties of existing objects.

UCS
Manages user coordinate systems.

System Variables

GFANG
Specifies the angle of a gradient fill.

GFCLR1
Specifies the color for a one-color gradient fill or the first color for a two-color gradient fill.

GFCLR2
Specifies the second color for a two-color gradient fill.

GFCLRLUM
Controls the tint or shade level in a one-color gradient fill.
GFCLRSTATE
    Specifies whether a gradient fill uses one color or two colors.

GFNAME
    Specifies the pattern of a gradient fill.

GFSHIFT
    Specifies whether the pattern in a gradient fill is centered or is shifted up
    and to the left.

HPANG
    Sets the angle for new hatch patterns.

HPANNOTATIVE
    Controls whether a new hatch pattern is annotative.

HPASSOC
    Controls whether hatches and fills are associative.

HPBACKGROUNDCOLOR
    Controls the background color for hatch patterns.

HPBOUND
    Controls the object type created by HATCH and BOUNDARY.

HPBOUNDRETAIN
    Controls whether boundary objects are created for new hatches and fills.

HPCOLOR
    Sets a default color for new hatches.

HPDOUBLE
    Specifies hatch pattern doubling for user-defined patterns.

HPDRAWORDER
    Controls the draw order of hatches and fills.

HPINHERIT
    Controls whether to inherit the hatch origin when using the Inherit Properties
    option in HATCH and HATCHEDIT.
HPISLANDDETECTION
Controls how islands within the hatch boundary are treated.

HPISLANDDETECTIONMODE
Controls whether internal closed boundaries, called islands, are detected.

HPLAYER
Specifies a default layer for new hatches and fills.

HPMAXLINES
Sets the maximum number of hatch lines that are generated in a hatch operation.

HPNAME
Sets the default hatch pattern name.

HPOBJWARNING
Sets the number of hatch boundary objects that can be selected before displaying a warning message.

HPINHERIT
Controls whether to inherit the hatch origin when using the Inherit Properties option in HATCH and HATCHEDIT.

HPORIGIN
Sets the hatch origin point for new hatch patterns relative to the current user coordinate system.

HPORIGINMODE
Controls how the default hatch origin point is determined.

HPQUICKPREVIEW
Controls whether to display a preview when specifying internal points for a hatch.

HPQUICKPREVTIMEOUT
Sets the maximum duration that the hatch preview should be displayed before the preview is canceled.

HPSCALE
Sets the hatch pattern scale factor.
HPSEPARATE
Controls whether a single hatch object or separate hatch objects are created when operating on several closed boundaries.

HPSPACE
Sets the hatch pattern line spacing for user-defined patterns.

HPTRANSPARENCY
Sets the default transparency for new hatches and fills.

MIRRHATCH
Controls how MIRROR reflects hatch patterns.

Control the Hatch Origin Point
Each hatch pattern is aligned with an origin point. Changing the origin point shifts the pattern.

By default, hatch patterns are aligned with the origin point of the user coordinate system. However, sometimes you need to move the origin point of the hatch object. For example, if you create a brick pattern, you can start with a complete brick in the lower-left corner of the hatched area by specifying a new origin point.

The hatch origin and its behavior depend on settings in the user interface that control the HPORIGIN, HPORIGINMODE, and HPINHERIT system variables. Alternatively, you can control hatch patterns by changing the location and orientation of the user coordinate system.

See also:
- Modify Hatch Alignment, Scale, and Rotation on page 698
Quick Reference

Commands

HATCH
Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

HATCHEDIT
Modifies an existing hatch or fill.

PROPERTIES
Controls properties of existing objects.

UCS
Manages user coordinate systems.

System Variables

HPINHERIT
Controls whether to inherit the hatch origin when using the Inherit Properties option in HATCH and HATCHEDIT.

HPORIGIN
Sets the hatch origin point for new hatch patterns relative to the current user coordinate system.

HPORIGINMODE
Controls how the default hatch origin point is determined.

HPQUICKPREVIEW
 Controls whether to display a preview when specifying internal points for a hatch.

HPQUICKPREVTIMEOUT
Sets the maximum duration that the hatch preview should be displayed before the preview is canceled.

HPSCALE
Sets the hatch pattern scale factor.
**HPSEPARATE**

Controls whether a single hatch object or separate hatch objects are created when operating on several closed boundaries.

**HPSPACE**

Sets the hatch pattern line spacing for user-defined patterns.

---

**Control the Scale of Hatch Patterns**

The scale of hatch patterns can be set individually, or it can be set automatically based on the scale of each layout viewport.

- If you create hatch patterns exclusively for a single view or at a constant scale, you can set the current hatch scale manually in the interface or with the HPScale system variable.

- If you work with layout viewports in different scales, you can apply scale factors automatically by making them annotative. This method is more efficient than creating duplicate hatch pattern objects with different scale factors. For more information about using annotative scaling, see Create Annotative Hatches on page 669.

**NOTE** To prevent accidental creation of an enormous number of hatch lines, the maximum number of hatch lines created in a single hatch operation is limited. This limit prevents memory and performance problems. However, you can change the maximum number of hatch lines with the HPMAXLINES system variable.

---

**See also:**

- Scale Annotations on page 656
- Modify Hatch Alignment, Scale, and Rotation on page 698
- Create Annotative Hatches on page 669

---

**Quick Reference**

**Commands**

HATCH

Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.
HATCHEDIT
Modifies an existing hatch or fill.

MATCHPROP
Applies the properties of a selected object to other objects.

PROPERTIES
Controls properties of existing objects.

System Variables
HPANNOTATIVE
Controls whether a new hatch pattern is annotative.

HPINHERIT
Controls whether to inherit the hatch origin when using the Inherit Properties option in HATCH and HATCHEDIT.

HPMAXLINES
Sets the maximum number of hatch lines that are generated in a hatch operation.

HPOBJWARNING
Sets the number of hatch boundary objects that can be selected before displaying a warning message.

HPINHERIT
Controls whether to inherit the hatch origin when using the Inherit Properties option in HATCH and HATCHEDIT.

HPSCALE
Sets the hatch pattern scale factor.

HPSPACE
Sets the hatch pattern line spacing for user-defined patterns.

Set Property Overrides for Hatches and Fills
Control the default color, layer, and transparency of hatch objects separately from other objects.
Hatch objects have an additional capability that is not available with other types of objects. You can specify which layer, color, and transparency settings will be automatically applied to each new hatch object, regardless of the current property settings. This can save you time.

For example, you can specify that all new hatch objects are automatically created on a specified layer regardless of the current layer setting.

**NOTE** If you do not want to override the current property settings, select Use Current for the hatch’s layer, color, and transparency settings.

See also:

- Modify Hatch Properties on page 697
- Control How Overlapping Objects Are Displayed on page 196

**Quick Reference**

**Commands**

**HATCH**

Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

**HATCHEDIT**

Modifies an existing hatch or fill.

**HATCHTOBACK**

Sets the draw order for all hatches in the drawing to be behind all other objects.

**MATCHPROP**

Applies the properties of a selected object to other objects.

**PROPERTIES**

Controls properties of existing objects.

**System Variables**

**HPANNOTATIVE**

Controls whether a new hatch pattern is annotative.
**HPASSOC**
Controls whether hatches and fills are associative.

**HPBACKGROUND COLOR**
Controls the background color for hatch patterns.

**HPCOLOR**
Sets a default color for new hatches.

**HPDRAWORDER**
Controls the draw order of hatches and fills.

**HPINHERIT**
Controls whether to inherit the hatch origin when using the Inherit Properties option in HATCH and HATCHEDIT.

**HPLAYER**
Specifies a default layer for new hatches and fills.

**HPSCALE**
Sets the hatch pattern scale factor.

**HPSEPARATE**
Controls whether a single hatch object or separate hatch objects are created when operating on several closed boundaries.

**HPTRANSPARENCY**
Sets the default transparency for new hatches and fills.

**MIRRHATCH**
Controls how MIRROR reflects hatch patterns.

---

**Control the Display of Hatch Boundaries**

Hide or remove boundary objects to create hatches without borders.

To create hatches that have no boundary objects, do one of the following:

- Erase the boundary objects of an existing hatch.
- Trim an existing hatch to objects that cross the edges of the hatch. After trimming, erase the objects.
Define hatch boundary points with the Draw option of the \texttt{-HATCH} command.

To hide a hatch's boundary objects, assign the boundary objects to a different layer than the hatch object, and then turn off or freeze the layer of the boundary objects. This method maintains hatch associativity.

See also:

- Reshape a Hatch or Fill on page 699

Quick Reference

Commands

- \texttt{-HATCH}
  - conref to \texttt{-Hatch blurb}

Properties

- Controls properties of existing objects.

System Variables

\texttt{HPANG}

- Sets the angle for new hatch patterns.

\texttt{HPBOUNDRETAIN}

- Controls whether boundary objects are created for new hatches and fills.

\texttt{HPCOLOR}

- Sets a default color for new hatches.
HPDOUBLE
Specifies hatch pattern doubling for user-defined patterns.

HPMAXLINES
Sets the maximum number of hatch lines that are generated in a hatch operation.

HPNAME
Sets the default hatch pattern name.

HPOBJWARNING
Sets the number of hatch boundary objects that can be selected before displaying a warning message.

HPINHERIT
Controls whether to inherit the hatch origin when using the Inherit Properties option in HATCH and HATCHEDIT.

HPORIGIN
Sets the hatch origin point for new hatch patterns relative to the current user coordinate system.

HPORIGINMODE
Controls how the default hatch origin point is determined.

HPSCALE
Sets the hatch pattern scale factor.

HPSPACE
Sets the hatch pattern line spacing for user-defined patterns.

HPTRANSPARENCY
Sets the default transparency for new hatches and fills.

Control the Draw Order of Hatches and Fills
Specify the draw order for a hatch object to control whether it is displayed behind or in front of the hatch boundary, or behind or in front of all other objects.

This behavior is controlled by the HPDRAWORDER system variable.
In drawings that contain many hatch objects, use the HATCHTOBACK command to display all hatch objects behind all other objects.

Quick Reference

Commands
HATCHTOBACK
Sets the draw order for all hatches in the drawing to be behind all other objects.

System Variables
HPDRAWORDER
Controls the draw order of hatches and fills.

Modify Hatches and Fills
Modify hatch properties and boundaries, or re-create the boundaries hatch objects.

Modify Hatch Properties
Modify the properties of hatch objects directly or copy them from another hatch object.

The following tools are available for modifying hatch properties:

- Hatch Edit dialog box. Access the dialog box with HATCHEDIT.
- Properties Inspector.
- Hatch shortcut menu. Access the menu by right-clicking a hatch object.
- Hatch dynamic menu. Access the menu by hovering over the control grip on a selected hatch.
- Command line. Enter -HATCHEDIT.

Copy the properties of one hatch to another using the following methods:

- Inherit Properties button in the Hatch Edit dialog box. Copy all hatch-specific properties.
■ **Match Properties command.** Use MATCHPROP to copy general properties and hatch-specific properties, with the exception of the hatch origin.

See also:
■ **Control How Overlapping Objects Are Displayed** on page 196

**Quick Reference**

**Commands**

HATCHEDIT
---
Modifies an existing hatch or fill.

HATCHTOBACK
---
Sets the draw order for all hatches in the drawing to be behind all other objects.

MATCHPROP
---
Applies the properties of a selected object to other objects.

PROPERTIES
---
Controls properties of existing objects.

**System Variables**

HPINHERIT
---
Controls whether to inherit the hatch origin when using the Inherit Properties option in HATCH and HATCHEDIT.

**Modify Hatch Alignment, Scale, and Rotation**

Shift, scale, or rotate hatch patterns to align them with existing objects.

To shift a hatch pattern, relocate the origin point of the hatch object. The same tools in the user interface as listed in **Modify Hatch Properties** on page 697 include options for specifying a new origin point, specifying a different rotation angle, and changing the scale of the hatch pattern.

In some cases, it might be easier to move or rotate the user coordinate system to align with existing objects, and then recreate the hatch.
Quick Reference

Commands

HATCHEDIT
Modifies an existing hatch or fill.

MATCHPROP
Applies the properties of a selected object to other objects.

PROPERTIES
Controls properties of existing objects.

System Variables

HPINHERIT
Controls whether to inherit the hatch origin when using the Inherit Properties option in HATCH and HATCHEDIT.

Reshape a Hatch or Fill

Reshape an associative hatch by modifying the boundary objects. Reshape a nonassociative hatch by modifying the hatch object.

Modify the Extents of Associative Hatches and Fills

If you modify the boundary objects of an associative hatch, and the result maintains a closed boundary, the associated hatch object is automatically updated. If the changes result in an open boundary, the hatch loses its associativity with the boundary objects, and the hatch remains unchanged.

When you select an associative hatch object, it displays a circular grip, called the control grip, at the center of the hatch extents. Hover over the control grip to display a shortcut menu with several hatch options, or right-click to display additional options.
You can also change the hatch object by editing the grips of the associated boundary objects. To easily select all of the objects in a complex boundary, use the Display Boundary Objects option.

If the boundary object is a polyline or spline, multi-functional grips are displayed. For more information, see Modify Objects with Multi-Functional Grips on page 316.

**Modify the Extents of Non-associative Hatches and Fills**

When you select a non-associative hatch, multi-functional grips are displayed on the hatch. Use these grips to modify the hatch extents and some several hatch properties.

When you hover over a grip on a nonassociative hatch object, a grip menu displays several edit options based on the type of grip. For example, a linear segment grip has an option to convert the segment to an arc, or to add a vertex.
NOTE For drastic changes, you can use TRIM to reduce the area covered by a hatch object, or EXPLODE to disassemble a hatch into its component objects.

See also:
■ Modify Objects with Multi-Functional Grips on page 316

Quick Reference

Commands
EXPLODE
Breaks a compound object into its component objects.

TRIM
Trims objects to meet the edges of other objects.

System Variables
GRIPS
Controls the display of grips on selected objects.

Re-create the Boundary of a Hatch or Fill
Create a new boundary object for a non-associative or an unbounded hatch or fill.

Use the Recreate Boundary option to generate a closed polyline or a region object around a selected hatch or fill. You can also specify that the new boundary object is associated with the hatch.
Quick Reference

Commands

HATCHEDIT
Modifies an existing hatch or fill.

System Variables

HPBOUND
Controls the object type created by HATCH and BOUNDARY.

HPBOUNDRETAIN
Controls whether boundary objects are created for new hatches and fills.

HPSEPARATE
Controls whether a single hatch object or separate hatch objects are created when operating on several closed boundaries.

Create a Blank Area to Cover Objects

Create a polygonal area, called a wipeout to mask underlying objects with the current background color.

A wipeout object covers existing objects with a blank area to make room for notes or to mask details. This area is defined by the wipeout frame, which you can turn on for editing, and turn off for plotting.

Use the WIPEOUT command both for creating a wipeout object, and for controlling whether wipeout frames are displayed or hidden in the drawing.

If a polyline is used to create a wipeout object, the polyline must be closed, contain line segments only, and have zero width.
Use Wipeout Objects on a Layout

You can create wipeout objects on a layout in paper space to mask objects in model space. However, in the Page Settings - Advanced dialog box, under Print Options, the Plot Paperspace Last option must be cleared before you print to ensure that the wipeout object is printed correctly.

See also:

■ Control How Overlapping Objects Are Displayed on page 196

Quick Reference

Commands

WIPEOUT

Creates a wipeout object, and controls whether wipeout frames are displayed in the drawing.
You can create and modify several types of text, including text with leaders. You can control most text style settings by defining text styles.

**Overview of Notes and Labels**

You can create text in various ways. For short, simple entries, use single-line text. For longer entries with internal formatting, use multiline text (mtext).

Although all entered text uses the current text style, which establishes the default font and format settings, you can use several methods to customize the text appearance. There are several tools that can change text scale and justification, find and replace text, and check for spelling errors.

Text that is included in a dimension or tolerance is created using the dimensioning commands. You can also create multiline text with leaders.

**Quick Reference**

**Commands**

DDEDIT

Edits single-line text, dimension text, attribute definitions, and feature control frames.

MLEADER

Creates a multileader object.

MTEXT

Creates a multiline text object.
Spell
Checks spelling in a drawing.

Style
Creates, modifies, or specifies text styles.

Text
Creates a single-line text object.

System Variables

DIMASZ
Controls the size of dimension line and leader line arrowheads.

DIMLDRBLK
Specifies the arrow type for leaders.

Create Text
You can create text using several methods, depending on your needs.

See also:
- Use Fields in Text on page 732

Overview of Creating Text
The text you add to your drawings conveys a variety of information. It may be a complex specification, title block information, a label, or even part of the drawing.

Single-Line Text
For short entries that do not require multiple fonts or lines, create single-line text. Single-line text is most convenient for labels.

Multiline Text
For long, complex entries, create multil ine, or paragraph text. Multiline text consists of any number of text lines or paragraphs that fit within a width you specify; it can extend vertically to an indefinite length.
Regardless of the number of lines, each set of paragraphs created in a single editing session forms a single object, which you can move, rotate, erase, copy, mirror, or scale.

There are more editing options for multiline text than there are for single-line text. For example, you can apply underlining, fonts, color, and text height changes to individual characters, words, or phrases within a paragraph.

**Annotative Text**

Use text for notes and labels in your drawing. You create annotative text by using an annotative text style, which sets the height of the text on the paper.

For more information about creating and working with an annotative text, see [Create Annotative Text](#) on page 663.

**Quick Reference**

**Commands**

MTEXT

- Creates a multiline text object.

TEXT

- Creates a single-line text object.

**System Variables**

STYLE

- Creates, modifies, or specifies text styles.

TEXTED

- Specifies the user interface displayed for editing single-line text.

TEXTFILL

- Controls the filling of TrueType fonts while printing.
TEXTQLTY
Sets the resolution tessellation fineness of text outlines.

TEXTSIZE
Sets the default height for new text objects drawn with the current text style.

TEXTSTYLE
Sets the name of the current text style.

Create Single-Line Text
You can use single-line text to create one or more lines of text, where each text line is an independent object that you can relocate, reformat, or otherwise modify.

Use single-line text (TEXT) to create one or more lines of text, ending each line when you press Enter. Each text line is an independent object that you can relocate, reformat, or otherwise modify.

When you create single-line text, you assign a text style and set alignment. The text style sets the default characteristics of the text object. The alignment determines what part of the text character aligns with the insertion point. Use the TEXT command to enter the text in-place, or enter -text to type text at the Command prompt instead of in-place.

You can insert a field in single-line text. A field is text that is set up to display data that might change. When the field is updated, the latest value of the field is displayed.

The text styles used for single-line text are the same as those used for multiline text. When you create text, you assign an existing style by entering its name at the Style prompt. If you need to apply formatting to individual words and characters, use multiline text instead of single-line text.

You can also compress single-line text to fit between points that you specify. This option stretches or squeezes the text to fill the designated space.

The TEXTED system variable specifies the user interface displayed for editing single-line text.

Align Single-Line Text
As you create text, you can align it. That is, you can justify it with one of the alignment options shown in the following illustrations. Left alignment is the default. To left-align text, do not enter an option at the Justify prompt.
See also:

- **Use Fields in Text** on page 732

## Quick Reference

### Commands

**QTEXT**

Controls the display and plotting of text and attribute objects.
STYLE
  Creates, modifies, or specifies text styles.

TEXT
  Creates a single-line text object.

System Variables

FONTALT
  Specifies the alternate font to be used when the specified font file cannot be located.

FONTMAP
  Specifies the font mapping file to be used.

MIRRTEXT
  Controls how MIRROR reflects text.

QTEXTMODE
  Controls how text is displayed.

TEXTED
  Specifies the user interface displayed for editing single-line text.

TEXEVAL
  Controls how text strings entered with TEXT (using AutoLISP) or with -TEXT are evaluated.

TEXTFILL
  Controls the filling of TrueType fonts while printing.

TEXTQLTY
  Sets the resolution tessellation fineness of text outlines.

TEXTSIZE
  Sets the default height for new text objects drawn with the current text style.

TEXTSTYLE
  Sets the name of the current text style.
Create Multiline Text

A multiline text (mtext) object includes one or more paragraphs of text that can be manipulated as a single object.

Overview of Multiline Text

You can create a multiline text (mtext) object by entering or importing text.

You can create one or more paragraphs of multiline text (mtext) in the In-Place Text Editor. You can also type text at the Command prompt if you use -MTEXT. You can insert text from a file saved in ASCII or RTF format.

Before entering or importing text, you specify opposite corners of a text bounding box that defines the width of the paragraphs in the multiline text object. The length of the multiline text object depends on the amount of text, not the length of the bounding box. You can use grips to move or rotate a multiline text object.

**NOTE** Multiline text objects and imported text files are limited to 256 KB in size.

The In-Place Text Editor allows you to adjust the bounding box that defines the size of the multiline text object, as well as create and edit tabs and indents on the ruler along the top. The editor is transparent, as you create text, you can see whether the text overlaps other objects.

To turn off transparency while you work, right-click in the In-Place Text Editor and click Opaque Background from the shortcut menu. You can also make the background of the finished multiline text object opaque and set its color.

You can also insert fields in multiline text. A field is text that is set up to display data that might change. When the field is updated, the latest value of the field is displayed.

Text Style

Most characteristics of the text are controlled by the text style, which sets the default font and other options, such as line spacing, justification, and color. You can use the current text style or select a new one. The STANDARD text style is the default.

Within the multiline text object, you can override the current text style by applying formatting such as underlining, boldface, and different fonts to individual characters. You can also create stacked text, such as fractions or geometric tolerances and insert special characters, including Unicode characters, for TrueType fonts.
**NOTE** Not all SHX and TrueType text fonts support Unicode characters.

**Text Properties**

In the Properties Inspector 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.

**Quick Reference**

**Commands**

**MTEXT**
- Creates a multiline text object.

**QTEXT**
- Controls the display and plotting of text and attribute objects.

**STYLE**
- Creates, modifies, or specifies text styles.

**System Variables**

**MTEXTCOLUMN**
- Sets the default column setting for an mtext object.

**MTEXTFIXED**
- Sets the display size and orientation of multiline text in a specified text editor.

**MTJIGSTRING**
- Sets the content of the sample text displayed at the cursor location when the MTEXT command is started.
QTEXTMODE

Controls how text is displayed.

TEXTFILL

Controls the filling of TrueType fonts while printing.

TEXTQLTY

Sets the resolution tessellation fineness of text outlines.

TEXTSIZE

Sets the default height for new text objects drawn with the current text style.

TEXTSTYLE

Sets the name of the current text style.

**Justify Multiline Text**

Justification of multiline text objects controls both text alignment and text flow relative to the text insertion point.

Justification controls both text alignment and text flow relative to the text insertion point. Text is left-justified and right-justified with respect to the boundary rectangle that defines the text width. Text flows from the insertion point, which can be at the middle, the top, or the bottom of the resulting text object.

There are nine justification settings for multiline text.

If a single word is longer than the width of the paragraph, the word will extend beyond the paragraph boundary.
Quick Reference

Commands

PROPERTIES

Controls properties of existing objects.

Format Characters Within Multiline Text

You can override the text style and apply different formatting to individual words and characters within multiline text.

The format changes affect only the text you select; the current text style is not changed.

You can specify a different font and text height and apply boldface, italics, underlining, overlining, and color. You can also set an obliquing angle, change the space between characters, and make characters wider or narrower. The
Remove Formatting option on the In-Place Text Editor shortcut menu resets the character attributes of selected text to the current text style and text color.

The text height setting specifies the height of capitalized text. For more information about how height is calculated, see MTEXT.

**See also:**
- Work with Text Styles on page 738

**Quick Reference**

**Commands**

DDEDIT
- Edits single-line text, dimension text, attribute definitions, and feature control frames.

MTEXT
- Creates a multiline text object.

PROPERTIES
- Controls properties of existing objects.

STYLE
- Creates, modifies, or specifies text styles.

**System Variables**

TEXTFILL
- Controls the filling of TrueType fonts while printing.

TEXTSTYLE
- Sets the name of the current text style.

**Create Lists in Multiline Text**

You can create bulleted lists, lettered or numbered lists, or simple outlines in multiline text.

Lines of multiline text can be formatted as a list. When you add or delete an item, or move an item up or down a level, the list numbering automatically
adjusts. You can remove and reapply list formatting with the same method as used in most text editors.

**Use Automatic List Formatting**

By default, list formatting is applied to all text that looks like a list. Text that meets all the following criteria is considered to be a list:

- The line begins with one or more letters or numbers or a symbol.
- The letters or numbers is followed by punctuation.
- A space after the punctuation is created by pressing Tab.
- The text following the space is ended by Enter or Shift-Enter.

**NOTE** If you do not want list formatting applied to all text that fits the criteria, clear the Allow Bullets and Lists option. (Right-click in the In-Place Text Editor, click Bullets and Lists ➤ Allow Bullets and Lists.) When Allow Bullets and Lists is not checked, you cannot create new formatted lists in the multiline text object.

To create a list, use one of the following methods:

- Apply list formatting to new or selected text.
- Use Auto-list (on by default) and type the elements of a list.
- With Auto-list off, type the elements of a list and close and reopen the editor to convert the text to a list.

**Apply List Formatting**

When you apply list formatting, you can specify bullets, uppercase or lowercase letters, or numbers. Default settings are used for the type of list you choose. Letters or numbers are followed by a period. Nested lists use a double bullet, letter, or number. Items are indented based on the tab stops on the ruler in the In-Place Text Editor.

**Use Auto-list to Type a List**

When Auto-list is on, you can create a list as you type. You can use letters, numbers, or symbols.

For example, in the editor, enter \U+25CB, press Tab, and then enter some text. This creates a empty circle style bullet.
Not all symbols are available from the character map for a particular text font. However, if you specify the Unicode text directly (\U+25CB in this case), you can always get the bullet format of your choice.

**NOTE** Press Tab after you enter the Unicode text or symbol, or it will remain a separate character.

You can also paste a symbol from the Characters dialog box.

The following characters can be used as punctuation after the number or letter when you type a list but cannot be used as bullets:

<table>
<thead>
<tr>
<th>Character</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>.</td>
<td>Period</td>
</tr>
<tr>
<td>:</td>
<td>Colon</td>
</tr>
<tr>
<td>)</td>
<td>Close parenthesis</td>
</tr>
<tr>
<td>&gt;</td>
<td>Close angle bracket</td>
</tr>
<tr>
<td>]</td>
<td>Close square bracket</td>
</tr>
<tr>
<td>}</td>
<td>Close curly bracket</td>
</tr>
</tbody>
</table>

**Paste a List from Another Document**

If you copy a nested bulleted list (a list within a list) from a word processor and paste the list into a multiline text, the bullets that are displayed as empty circles might not be formatted like other bullets in multiline text. This is because the bullet might be a letter, such as o, instead of a bullet for nested bulleted lists. You can remove formatting from the nested list and reapply to change the bullets to double bullets.

**Quick Reference**

**Commands**

**DDEDIT**

Edits single-line text, dimension text, attribute definitions, and feature control frames.
MTEXT
Creates a multiline text object.

PROPERTIES
Controls properties of existing objects.

System Variables

TEXTFILL
Controls the filling of TrueType fonts while printing.

TEXTQLTY
Sets the resolution tessellation fineness of text outlines.

Indent Multiline Text and Use Tabs
You can control how paragraphs are indented in a multiline text (mtext) object. The ruler in the In-Place Text Editor shows the settings for the current paragraph.

Tabs and indents that you set before you start to enter text apply to the whole multiline text object. To apply different tabs and indents to individual paragraphs, click in a single paragraph or select multiple paragraphs and then change the settings.

Sliders on the ruler show indentation relative to the left side of the bounding box. The top slider indents the first line of the paragraph, and the bottom slider indents the other lines of the paragraph.

The long tick marks on the ruler show the default tab stops. If you click the ruler to set your own tabs, the ruler displays a small, L-shaped marker at each custom tab stop. You can delete a custom tab stop by dragging the marker off the ruler.
Quick Reference

Commands

DDEDIT
Edits single-line text, dimension text, attribute definitions, and feature control frames.

MTEXT
Creates a multiline text object.

PROPERTIES
Controls properties of existing objects.

System Variables

TEXTFILL
Controls the filling of TrueType fonts while printing.

TEXTQLTY
Sets the resolution tessellation fineness of text outlines.

Specify the Line Spacing Within Multiline Text

Line spacing for multiline text is the distance between the baseline (bottom) of one line of text and the baseline of the next line of text. The line space factor applies to the entire multiline text object, not to selected lines.

You can set the spacing increment to a multiple of single line spacing, or as an absolute distance. Single spacing is 1.66 times the height of the text characters.

The default line space style, At Least, automatically increases line spacing to accommodate characters that are too large to fit the line spacing you set for the multiline text object. Use the other line space style, Exactly, to line up text in tables.

To ensure that line spacing is identical in multiple multiline text objects, use Exactly and set the Line Space Factor to the same value in each multiline text object.

NOTE Using Exactly can cause text in lines located above or below lines with large font characters to overlap the larger characters.
Quick Reference

Commands
PROPERTIES
Controls properties of existing objects.

System Variables
TSPACEFAC
Controls the multiline text line-spacing distance measured as a factor of text height.
TSPACETYPE
Controls the type of line spacing used in multiline text.

Create Stacked Characters Within Multiline Text
Characters representing fractions and tolerances can be formatted to conform to several standards.

Stacked text refers to the fraction and tolerance formats applied to characters within multiline text object and multileaders.

Unstacked text: \(0.054 - 0.057\text{ DIA-2HOLES}\)
Tolerance stack: \(0.054 - 0.067\text{ DIA-2HOLES}\)
Diagonal fraction: \(1{\frac{1}{2}}\text{ Ø GALV.STL. POSTS}\)
Vertical fraction: \(1{\frac{3}{4}}\text{ Ø GALV.STL. POSTS}\)
You use special characters to indicate how selected text should be stacked.

- Slash (/) stacks text vertically, separated by a horizontal line.
- Pound sign (#) stacks text diagonally, separated by a diagonal line.
- Carat (^) creates a tolerance stack, which is stacked vertically and not separated by a line.

To stack characters manually within the In-Place Text Editor, select the text to be formatted, including the special stacking character, and right-click. From the shortcut menu, click Stack.

**Stack Numeric and Tolerance Characters Automatically**

You can specify that numeric characters entered before and after a slash, pound sign, or carat will stack automatically. For example, if you enter 1#3 followed by a nonnumeric character or space, the AutoStack Properties dialog box is displayed by default, and you can change the settings in the Stack Properties dialog box to specify your formatting preferences.

The automatic stacking feature applies only to numeric characters immediately before and after the slash, pound sign, and carat. For tolerance stacking, the +, -, and decimal character also stack automatically.

**See also:**
- Work with Text Styles on page 738

**Quick Reference**

**Commands**

MTEXT

Creates a multiline text object.

**System Variables**

TSTACKALIGN

Controls the vertical alignment of stacked text.

TSTACKSIZE

Controls the percentage of stacked text fraction height relative to selected text's current height.
Create and Edit Columns in Multiline Text

You can create and edit multiple columns using the In-Place Text Editor column options and column grips.

Multiple columns can be created and edited with the In-Place Text Editor and through grip editing. Editing columns using grips allows you the flexibility of seeing the changes as you make them.

Columns follow a few rules. All columns have equal width and equal gutters. A gutter is the space between columns. The height of columns remains constant unless more text than the column can accommodate is added, or you manually move the editing grip to adjust the column height.

Editing Columns in the In-Place Text Editor

When you are working with columns in the In-Place Text Editor, the columns will be in a frame. The ruler bar spans across all columns, but is only active for the current column.

Adding text to a column with an arbitrary height will not increase the column height even if text is already filling the column. Text will flow into another column.

You can also insert a column break to force text to start flowing into the next column. Anytime a column break is inserted, it is assumed that the current height of the column is fixed. To delete the break, highlight it and delete or use the Backspace key right after the break.

Editing Columns in the Properties Inspector

You will be able to select Static or Dynamic columns, turn off columns and change column and gutter width through the Properties Inspector palette. Changing column width in the palette will exhibit results similar to changing width using grips. The palette is the only place that you can also change gutter setting.
Quick Reference

Commands
MTEXT
   Creates a multiline text object.

System Variables
MTEXTCOLUMN
   Sets the default column setting for an mtext object.

Import Text from External Files
You can insert TXT or RTF text files created in word processors into your drawing by importing the text.

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 multilime text object, which you can edit and reformat. Text imported from a TXT file inherits the current text style. Text imported from an RTF file inherits the current text style name, but retains its original fonts and format. Imported text files are limited to 256 KB and must have a file extension of .txt or .rtf.

If you use the Clipboard to paste text from another application, the text is pasted as formatted or unformatted based on the original source. If you use the Clipboard to paste text from another drawing file, the text is inserted as a block reference, and it retains its original formatting.

Quick Reference

Commands
MTEXT
   Creates a multiline text object.

Create Leaders
You can create, modify and add content to a leader object.
Overview of Leader Objects

A leader object is a line or a spline with an arrowhead at one end and a multiline text object or block at the other.

In some cases, a short horizontal line, called a landing, connects text or blocks and feature control frames to the leader line.

The landing and leader line are associated with the multiline text object or block, so when the landing is relocated, the content and leader line move along with it.

When associative dimensioning is turned on and object snaps are used to locate the leader arrowhead, the leader is associated with the object to which the arrowhead is attached. If the object is relocated, the arrowhead is relocated, and the landing stretches accordingly.

NOTE The leader object should not be confused with the leader line that is automatically generated as part of a dimension line.

Quick Reference

Commands

LEADER

Creates a line that connects annotation to a feature.

MLEADER

Creates a multiline leader object.
QLEADER

Creates a leader and leader annotation.

System Variables

DIMASSOC

Controls the associativity of dimension objects and whether dimensions are exploded.

DIMGAP

Sets the distance around the dimension text when the dimension line breaks to accommodate dimension text.

DIMLDRBLK

Specifies the arrow type for leaders.

MLEADERSCALE

Sets the overall scale factor applied to multileader objects.

Create and Modify Leaders

A leader object typically consists of an arrowhead, an optional horizontal landing, a leader line or curve, and either a multiline text object or block.

You can create a leader line from any point or feature in a drawing and control its appearance as you draw. Leaders can be straight line segments or smooth spline curves.

A multileader object, or mleader, comprises a leader and a note. It can be created arrowhead first, tail first, or content first. If a multileader style has been used, then the multileader can be created from that style.

Multileader objects can contain multiple leader lines, each of which can have one or more segments, so that one note can point to multiple objects in your
drawing. You can modify the properties of leader segment in the Properties Inspector palette. Add leaders to or remove leaders from a multileader object with MLEADEREDIT.

Annotative multileaders containing multiple leader segments can have different head points in each scale representation. Horizontal landings and arrowheads can have different sizes, and landing gaps can have different distances, depending on the scale representation. The appearance of the horizontal landing within a multileader, as well as the type of leader line (straight or spline) and number of leader segments will remain the same in all scale representations. For more information, see Create Annotative Leaders and Multileaders on page 666.

You can use grips to modify the look of a multileader. Using grips, you can lengthen or shorten a landing or leader line, or move the entire leader object.

Arrange Leaders
Multileaders can be arranged to add order and consistency to your drawing. Multileader objects with blocks as content can be collected and attached to one landing line. Using MLEADERCOLLECT, multileaders can be collected horizontally, vertically, or within a specified area depending on your drawing needs.

Multileader objects can be sorted evenly along a specified line. Using MLEADERALIGN, selected multileaders can be aligned and evenly spaced as specified.

Associate Leaders with Objects
When associative dimensioning is turned on (DIMASSOC system variable), the leader arrowhead can be associated with a location on an object using an
object snap. If the object is relocated, the arrowhead remains attached to the object and the leader line stretches, but the multiline text remains in place.

See also:

■ Create Annotative Leaders and Multileaders on page 666

Quick Reference

Commands

DDEDIT
Edits single-line text, dimension text, attribute definitions, and feature control frames.

MLEADER
Creates a multileader object.

MLEADERALIGN
Aligns and spaces selected multileader objects.

MLEADERCOLLECT
Organizes selected multileaders that contain blocks into rows or columns, and displays the result with a single leader.

MLEADEREDIT
Adds leader lines to, or removes leader lines from, a multileader object.

MLEADERSTYLE
Creates and modifies multileader styles.

PROPERTIES
Controls properties of existing objects.

System Variables

DIMASSOC
Controls the associativity of dimension objects and whether dimensions are exploded.

DIMASZ
Controls the size of dimension line and leader line arrowheads.
DIMCLRD
Assigns colors to dimension lines, arrowheads, and dimension leader lines.

DIMGAP
Sets the distance around the dimension text when the dimension line breaks to accommodate dimension text.

DIMLDRBLK
Specifies the arrow type for leaders.

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

DIMTAD
Controls the vertical position of text in relation to the dimension line.

DIMTXTDIRECTION
Specifies the reading direction of the dimension text.

MLEADERSCALE
Sets the overall scale factor applied to multileader objects.

Work with Leader Styles
The appearance of a leader is controlled by its multileader style. You can use the default multileader style, STANDARD, or create your own multileader styles.

The multileader style can specify formatting for landing lines, leader lines, arrowheads, and content. For example, the STANDARD multileader style uses a straight leader line with a closed filled arrowhead and multiline text content.

NOTE Annotative blocks cannot be used as either content or arrowheads in multileader objects.

Once a multileader style has been defined, you can set it as the current multileader style to be used when the MLEADER command is invoked.
Quick Reference

Commands
MLEADERSTYLE
Creates and modifies multileader styles.

System Variables
CMLEADERSTYLE
Sets the name of the current multileader style.

Add Content to a Leader
Leaders can contain multiline text or blocks to label parts of your drawing.

Leaders Containing Multiline Text
Leaders can contain multiline text as content. Text can be inserted by default when creating a leader style. Text style, color, height, and alignment can be applied and modified in leader annotations. You can also offset a multiline text object by specifying a landing gap distance in the current leader style.

You can create annotative multileaders with text as content. The text content will be scaled according to the specified scale representation. Width, justification, attachment, and rotation settings for text content can be different depending on the specified scale representation. Actual text content cannot change with the scale representation.

There are several options for placing multiline text as content in a leader object.

Top of top line

Middle of top line
Leaders Containing Blocks

Multileaders can contain blocks as content by applying a multileader style that references a block in your drawing.
NOTE  Annotative blocks cannot be used as either content or arrowheads in multileader objects.

Blocks can be connected to a multileader by attaching the landing to a selected insertion point on the block. You can also connect a multileader to a center point on the selected block.

You can create annotative multileaders with blocks as content. The block content will be scaled according to the specified scale representation. Any attributes within the block content will not change with the scale representation. Non-annotative multileader objects can be scaled using the MLEADERSCALE system variable.

See also:
- Use Fields in Text on page 732

Quick Reference

Commands

DDEDIT
  Edits single-line text, dimension text, attribute definitions, and feature control frames.

DIMSTYLE
  Creates and modifies dimension styles.

MLEADER
  Creates a multileader object.

MTEXT
  Creates a multiline text object.

OPTIONS
  Customizes the program settings.

PROPERTIES
  Controls properties of existing objects.

PURGE
  Removes unused items, such as block definitions and layers, from the drawing.
QTEXT
Controls the display and plotting of text and attribute objects.

SPELL
Checks spelling in a drawing.

STYLE
Creates, modifies, or specifies text styles.

TEXT
Creates a single-line text object.

**System Variables**

**DIMGAP**
Sets the distance around the dimension text when the dimension line breaks to accommodate dimension text.

**FONTALT**
Specifies the alternate font to be used when the specified font file cannot be located.

**FONTMAP**
Specifies the font mapping file to be used.

**MLEADERSCALE**
Sets the overall scale factor applied to multileader objects.

**QTEXTMODE**
Controls how text is displayed.

**TEXTFILL**
Controls the filling of TrueType fonts while printing.

**TEXTQLTY**
Sets the resolution tessellation fineness of text outlines.

**Use Fields in Text**
A field is updatable text that is set up to display data that may change during the life cycle of the drawing. When the field is updated, the latest value of the field is displayed.
**Insert Fields**

A field is text that contains instructions to display data that you expect to change during the life cycle of the drawing.

When a field is updated, the latest data is displayed. For example, the value of the FileName field is the name of the file. If the file name changes, the new file name is displayed when the field is updated.

Fields can be inserted in any kind of text (except tolerances), including text in table cells, attributes, and attribute definitions. When any text command is active, Insert Field is available on the shortcut menu.

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.

**Change the Appearance of a Field**

The field text uses the same text style as the text object in which it is inserted. By default, fields are displayed with a light gray background that is not plotted (FIELDDISPLAY system variable).

Formatting options in the Insert Field dialog box control the appearance of the text that is displayed. The options that are available depend on the type of field. For example, the format for date fields includes options for displaying the day of the week and the time, and the format for named object fields includes capitalization options.

**Edit a Field**

A field is part of a text object and it can be edited from a text editor. The easiest way to edit a field is to double click the text object that contains the field and then, to display the Insert Field dialog box, double click the field. These operations are available on the shortcut menus as well.

If you no longer want to update a field, you can preserve the value that is currently displayed by converting the field to text.

The field expression, consisting of escape characters and a field code, is shown in the Insert Field dialog box but cannot be edited.
Quick Reference

Commands
FIELD
Creates a multiline text object with a field that can be updated automatically as the field value changes.
FIND
Finds the text that you specify, and can optionally replace it with other text.
INSERT
Inserts a block or drawing into the current drawing.
LIST
Displays property data for selected objects.
MTEXT
Creates a multiline text object.
SPELL
Checks spelling in a drawing.
TABLE
Creates an empty table object.
TABLEEXPORT
Exports data from a table object in CSV file format.
TABLESTYLE
Creates, modifies, or specifies table styles.
UPDATEFIELD
Manually updates fields in selected objects in the drawing.

System Variables
CTABLESTYLE
Sets the name of the current table style.
FIELDDISPLAY
Controls whether fields are displayed with a gray background.
FIELDEVAL

Controls how fields are updated.

**Update Fields**

When a field is updated, it displays the latest value. You can update fields individually or update all fields in one or more selected text objects.

You can also set fields to be updated automatically when the drawing is opened, saved, printed, and regenerated.

FIELDEVAL controls whether fields are updated automatically or on demand. The Date field cannot be updated automatically regardless of the setting of FIELDEVAL.

---

**NOTE**

When the DEMANDLOAD system variable is set to 2, fields cannot be updated until you use FIELD or UPDATEFIELD.

---

**Contextual Fields in Blocks and Xrefs**

Some fields are contextual; that is, their value is different depending on which space or layout they reside in. For example, because each layout can have a different page setup attached, the value displayed by the PlotOrientation field can be different in different layouts in the same drawing.

**List of contextual fields**

- DeviceName
- PageSetupName
- PaperSize
- PlotDate
- PlotOrientation
- PlotScale
- PlotStyleTable

For compatibility with previous releases, contextual fields in blocks and xrefs are not updated when you insert them in a drawing; instead, the field displays
the last cached value. Therefore, if you want to use a contextual field within a block, for example, a title block, you must insert the field as an attribute.

NOTE  The Block Placeholder, Hyperlink, and SheetSet Manager fields are not available in AutoCAD 2011 for Mac. The drawings created in AutoCAD that contain these fields can be opened and the cached value is displayed.

Compatibility with Previous Releases

When a drawing with fields is opened in AutoCAD 2004 or earlier, the fields are not updated; they display the value last displayed in the drawing before it was opened. If no changes are made to a field, it is updated normally when it is reopened in a release that supports fields.

See also:
- Work with AutoCAD Drawings in AutoCAD LT on page 934

Quick Reference

Commands

FIELD
  Creates a multiline text object with a field that can be updated automatically as the field value changes.

UPDATEFIELD
  Manually updates fields in selected objects in the drawing.

System Variables

FIELDDISPLAY
  Controls whether fields are displayed with a gray background.

FIELDVAL
  Controls how fields are updated.

Use Hyperlinks in Fields

  The Hyperlink field assigns a hyperlink to any piece of text.
Hyperlinks are used to reference a URL, a file on disk, a saved view in a drawing, or a layout in a drawing file. Hyperlinks that are in a drawing created with the Windows version of AutoCAD cannot be accessed with AutoCAD for the Mac. The hyperlink is retained as long as the field is left unchanged.

**Quick Reference**

**Commands**

FIELD
Creates a multiline text object with a field that can be updated automatically as the field value changes.

FIND
Finds the text that you specify, and can optionally replace it with other text.

INSERT
Inserts a block or drawing into the current drawing.

LIST
Displays property data for selected objects.

MTEXT
Creates a multiline text object.

SPELL
Checks spelling in a drawing.

TABLE
Creates an empty table object.

TABLEEXPORT
Exports data from a table object in CSV file format.

TABLESTYLE
Creates, modifies, or specifies table styles.

UPDATEFIELD
Manually updates fields in selected objects in the drawing.
System Variables

CTABLESTYLE
Sets the name of the current table style.

FIELDDISPLAY
Controls whether fields are displayed with a gray background.

FIELDVAL
Controls how fields are updated.

Work with Text Styles
When you enter text into your drawing, the current text style determines the text font, size, angle, orientation, and other text characteristics.

Overview of Text Styles
All text in a drawing has a text style associated with it. When you enter text, the program uses the current text style.

The current text style sets the font, size, obliquing angle, orientation, and other text characteristics. If you want to create text using a different text style, you can make another text style current. The table shows the settings for the STANDARD text style.

The settings for the current text style are displayed at the Command prompts. You can use or modify the current text style or create and load a new text style. Once you've created a text style, you can modify its characteristics, change its name, or delete it when you no longer need it.

Create and Modify Text Styles
Except for the default STANDARD text style, you must create any text style that you want to use.

Text style names can be up to 255 characters long. They can contain letters, numbers, and the special characters dollar sign ($), underscore (_), and hyphen (-). If you don’t enter a text style name, the text styles are automatically named Stylen, where n is a number that starts at 1.

You can modify an existing text style in the Text Style dialog box by changing the settings. You can also update existing text of that text style to reflect the changes.
Certain style settings affect multiline and single-line text objects differently. For example, changing the Upside Down and Backwards options has no effect on multiline text objects. Changing Width Factor and Obliquing options has no effect on single-line text.

If you rename an existing text style, any text using the old name assumes the new text style name.

You can remove unreferenced text styles from your drawing with PURGE or by deleting the text styles from the Text Styles dialog box. The STANDARD text style cannot be removed.

### Change Text Style

When you change the text style of a multiline text object, the updated settings are applied to the entire object, and some formatting of individual characters might not be retained. The following table describes the effects of text style change on character formatting.

<table>
<thead>
<tr>
<th>Formatting</th>
<th>Retained?</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bold</td>
<td>No</td>
</tr>
<tr>
<td>Color</td>
<td>Yes</td>
</tr>
<tr>
<td>Font</td>
<td>No</td>
</tr>
<tr>
<td>Height</td>
<td>No</td>
</tr>
<tr>
<td>Italic</td>
<td>No</td>
</tr>
<tr>
<td>Stacking</td>
<td>Yes</td>
</tr>
<tr>
<td>Underlining</td>
<td>Yes</td>
</tr>
</tbody>
</table>

### Annotative Text Styles

Use text for notes and labels in your drawing. You create annotative text by using an annotative text style, which sets the height of the text on the paper.

For more information about creating and working with an annotative text, see Create Annotative Text on page 663.

**See also:**

- Scale Annotations on page 656
Quick Reference

Commands

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

STYLE
Creates, modifies, or specifies text styles.

System Variables

FONTALT
Specifies the alternate font to be used when the specified font file cannot be located.

FONTMAP
Specifies the font mapping file to be used.

TEXTSIZE
Sets the default height for new text objects drawn with the current text style.

TEXTSTYLE
Sets the name of the current text style.

Assign Text Fonts

You can assign a text font as part of the text style definition. Several factors depend on the type of text you are working with.

Overview of Assigning Text Fonts

Fonts define the shapes of the text characters that make up each character set. You can use TrueType fonts in addition to compiled SHX fonts.

A single font can be used by more than one text style. If your company has a standard font, you can modify other text style settings to create a set of text styles that use this standard font in different ways.
You can assign a font to a text style by selecting a font file from the list in the Text Style dialog box.

**Quick Reference**

**Commands**

**STYLE**

Creates, modifies, or specifies text styles.

**System Variables**

**FONTALT**

Specifies the alternate font to be used when the specified font file cannot be located.

**FONTMAP**

Specifies the font mapping file to be used.

**Use TrueType Fonts**

Several factors affect the display of TrueType fonts in a drawing.

TrueType fonts always appear filled in your drawing; however, when you plot, the TEXTFILL system variable controls whether the fonts are filled. By default TEXTFILL is set to 1 to plot the filled-in fonts.

The In-Place Text Editor can display only fonts that are recognized by the operating system. Because SHX fonts are not recognized by the operating system, a TrueType equivalent is supplied in the In-Place Text Editor when you select an SHX or any other non-TrueType font for editing.

**See also:**

- Set Text Height on page 746

**Quick Reference**

**Commands**

**QTEXT**

Controls the display and plotting of text and attribute objects.
STYLE
Creates, modifies, or specifies text styles.

System Variables
QTEXTMODE
Controls how text is displayed.
TEXTFILL
Controls the filling of TrueType fonts while printing.
TEXTQLTY
Sets the resolution tessellation fineness of text outlines.

Use Text Fonts for International Work
Several factors affect your choosing, entering, and displaying international text in a drawing.

The program supports the Unicode character-encoding standard. An SHX font encoded using the Unicode standard font can contain many more characters than are defined in your system; therefore, to use a character not directly available from the keyboard, you can enter the escape sequence \U+nnnn, where nnnn represents the Unicode hexadecimal value for the character.

Beginning with AutoCAD 2007, all SHX shape fonts are encoded with the Unicode standard with the exception of Asian sets, or more commonly known as Big Fonts. When choosing a text font for international work, you can use either a TrueType Font or a Big Font.

Asian Big Font SHX Files
Asian alphabets contain thousands of non-ASCII characters. To support such text, the program provides a special type of shape definition known as a Big Font file. You can set a style to use both regular and Big Font files.

<table>
<thead>
<tr>
<th>Asian Language Big Fonts Included in the Product</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Font File Name</strong></td>
</tr>
<tr>
<td>@extfont2.shx</td>
</tr>
<tr>
<td>bigfont.shx</td>
</tr>
</tbody>
</table>

742 | Chapter 27  Notes and Labels
## Asian Language Big Fonts Included in the Product

<table>
<thead>
<tr>
<th>Font File Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>chineset.shx</td>
<td>Traditional Chinese font</td>
</tr>
<tr>
<td>extfont1.shx</td>
<td>Japanese extended font, level 1</td>
</tr>
<tr>
<td>extfont2.shx</td>
<td>Japanese extended font, level 2</td>
</tr>
<tr>
<td>gbcbig.shx</td>
<td>Simplified Chinese font</td>
</tr>
<tr>
<td>whgd.txt.shx</td>
<td>Korean font</td>
</tr>
<tr>
<td>whgt.txt.shx</td>
<td>Korean font</td>
</tr>
<tr>
<td>whgtxt.txt.shx</td>
<td>Korean font</td>
</tr>
<tr>
<td>whgtxttxt.shx</td>
<td>Korean font</td>
</tr>
</tbody>
</table>

When you specify fonts using -STYLE, the assumption is that the first name is the normal font and the second (separated by a comma) is the Big Font. If you enter only one name, it's assumed that it is the normal font and any associated Big Font is removed. By using leading or trailing commas when specifying the font file names, you can change one font without affecting the other, as shown in the following table.

### Specifying fonts and Big Fonts at the Command prompt

<table>
<thead>
<tr>
<th>Enter this ...</th>
<th>To specify this ...</th>
</tr>
</thead>
<tbody>
<tr>
<td>[font name],[big font name]</td>
<td>Both normal fonts and Big Fonts</td>
</tr>
<tr>
<td>[font name],</td>
<td>Only a normal font (Big Font unchanged)</td>
</tr>
<tr>
<td>,[big font name]</td>
<td>Only a Big Font (normal font unchanged)</td>
</tr>
<tr>
<td>[font name]</td>
<td>Only a normal font (Big Font, if any, removed)</td>
</tr>
<tr>
<td>ENTER (null response)</td>
<td>No change</td>
</tr>
</tbody>
</table>

**NOTE** Long file names that contain commas as font file names are not accepted. The comma is interpreted as a separator for an SHX font-Big Font pair.
See also:

- Substitute Fonts on page 744

Quick Reference

Commands

STYLE

Creates, modifies, or specifies text styles.

Substitute Fonts

A font used in a drawing but that is not currently available on your system is automatically substituted with another font.

The program accommodates a font that is not currently on your system by substituting another font.

Specify an Alternate Font

If your drawing specifies a font that is not currently on your system, the font designated as your alternate font is automatically substituted. By default, the simplex.shx file is used. If you want to specify a different font, enter the alternate font file name by changing the FONTALT system variable. If you use a text style that uses a Big Font (or Asian Set), you can map it to another font using the FONTALT system variable. This system variable uses a default font file pair: txt.shx and bigfont.shx. For more information, see Use Text Fonts for International Work on page 742.

In previous releases, you could display PostScript® fonts in the drawing. Because later releases cannot display PostScript fonts, Autodesk has supplied TrueType font equivalents. These PostScript fonts are mapped to the equivalent TrueType fonts in a font mapping file. Additionally, when a TrueType font is not available, you can specify a different TrueType font, making sure that the fonts are similar to avoid text length or wrapping problems.

If the default font does not support the characters you enter using the In-Place Text Editor (MTEXT command), an alternative font is substituted.

CIF or MIF codes entered with the In-Place Text Editor or with the TEXT command are now automatically converted to display the actual characters.
Edit the Font Mapping File

A font mapping file is a list of text fonts and their substitutes. If a text font used in a drawing cannot be located, another text font is substituted for the missing font using a font mapping file.

Each line in the font mapping file contains the name of a font file (with no file extension or path) followed by a semicolon (;) and the name of the substitute font file. The substitute file name includes a file extension such as .ttf.

A font mapping file is an ordinary ASCII text file with a .fmp extension. The default font mapping file is acad.fmp. You can change the font assignments in a font mapping file using any ASCII text editor.

For example, you could use the following entry in a font map file to specify that the timesnr.pfb font file be substituted with the times.ttf font file:

timesnr;times.ttf

The following table shows the font substitution rules used if a font file cannot be located when a drawing is opened.

<table>
<thead>
<tr>
<th>File extension</th>
<th>First mapping order</th>
<th>Second mapping order</th>
<th>Third mapping order</th>
<th>Fourth mapping order</th>
</tr>
</thead>
<tbody>
<tr>
<td>.ttf</td>
<td>Use font mapping table</td>
<td>Use font defined in text style</td>
<td>Windows substitutes a similar font</td>
<td></td>
</tr>
<tr>
<td>.shx</td>
<td>Use font mapping table</td>
<td>Use font defined in text style</td>
<td>Use FONTALT</td>
<td>Prompt for new font</td>
</tr>
<tr>
<td>.pfb</td>
<td>Use font mapping table</td>
<td>Use FONTALT</td>
<td>Prompt for new font</td>
<td></td>
</tr>
</tbody>
</table>

Display Proxy Fonts

For third-party or custom SHX fonts that have no TrueType equivalent, one of several different TrueType fonts called proxy fonts is substituted. In the In-Place Text Editor, proxy fonts look different from the fonts they represent to indicate that the proxy fonts are substitutions for the fonts used in the drawing.

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.
Quick Reference

Commands
MTEXT
Creates a multiline text object.

OPTIONS
Customizes the program settings.

System Variables
FONTALT
Specifies the alternate font to be used when the specified font file cannot be located.

FONTMAP
Specifies the font mapping file to be used.

Set Text Height
Text height determines the size in drawing units of the letters in the font you are using.

The exception is TrueType fonts: the value usually represents the size of the uppercase letters.

If you specify a fixed height as part of a text style, the Height prompt is bypassed when you create single-line text. When the height is set to 0 in the text style, you are prompted for the height each time you create single-line text. Set the value to 0 if you want to specify the height as you create text.

TrueType Fonts
For TrueType fonts, the value specified for text height represents the height of a capital letter plus an ascent area reserved for accent marks and other marks used in non-English languages. The relative portion of text height that is assigned to capital letters and ascent characters is determined by the font designer at the time the font is designed; consequently, it varies from font to font.

In addition to the height of a capital letter and the ascent area that make up the text height specified by the user, TrueType fonts have a descent area for
portions of characters that extend below the text insertion line, for example, 
y, j, p, g, and q.

When you apply a text height override to all text in the editor, the entire multiline text object is scaled, including its width.

Quick Reference

Commands
STYLE
Creates, modifies, or specifies text styles.

System Variables
TEXTSIZE
Sets the default height for new text objects drawn with the current text style.
TEXTSTYLE
Sets the name of the current text style.

Set Text Obliquing Angle

The obliquing angle determines the forward or backward slant of the text. The angle represents the offset from 90 degrees.

Entering a value between -85 and 85 makes the text oblique. A positive obliquing angle slants text to the right. A negative obliquing angle slants text to the left.
Quick Reference

Commands
STYLE
Creates, modifies, or specifies text styles.

Set Horizontal or Vertical Text Orientation

Text can be vertical or horizontal. Text can have a vertical orientation only if the associated font supports dual orientation.

Lines of text are oriented to be vertical or horizontal. Text can have a vertical orientation only if the associated font supports dual orientation. You can create more than one line of vertical text. Each successive text line is drawn to the right of the preceding line. The normal rotation angle for vertical text is 270 degrees.

NOTE Vertical orientation is not supported for TrueType fonts and symbols.

Vertical Text for Asian Languages

- **SHX fonts.** Text can be created with SHX fonts and Big Fonts for vertical display in the same way as for previous releases. For best results, use the single-line TEXT command, not MTEXT. You can select a vertical style in the Text Style dialog box.

- **TrueType fonts.** You still select fonts starting with the @ sign, but now the text is automatically rotated 270 degrees. (In AutoCAD 2005 and earlier releases, you had to manually rotate this text.) Vertical cursor movement is now supported for vertical text.
Quick Reference

Commands

STYLE

Creates, modifies, or specifies text styles.

Change Text

You can change text content, formatting, and properties such as scale and justification.

Overview of Changing Text

Text, whether created with TEXT, MTEXT, or MLEADER can be modified like any other object.

You can move, rotate, erase, and copy it. You can change text properties in the Properties Inspector palette.

You can also edit the contents of existing text and create a mirror image of it. The MIRRTEXT system variable controls whether text is also reversed when you mirror objects in your drawing.

Quick Reference

Commands

DDEDIT

Edits single-line text, dimension text, attribute definitions, and feature control frames.

FIND

Finds the text that you specify, and can optionally replace it with other text.

PROPERTIES

Controls properties of existing objects.
System Variables

MIRRTEXT
Controls how MIRROR reflects text.

TEXTFILL
Controls the filling of TrueType fonts while printing.

TEXTQLTY
Sets the resolution tessellation fineness of text outlines.

Change Single-Line Text

You can change the contents, formatting and properties of single-line text.

You can change single-line text with DDEDIT and PROPERTIES. Use DDEDIT when you need to change only the content of the text, not the formatting or properties of the text object. Use PROPERTIES when you want to change content, text style, location, orientation, size, justification, and other properties.

Text objects also have grips for moving, scaling, and rotating. A text object has grips at the lower-left corner of the baseline and at the alignment point.

The effect of a command depends on which grip you choose.
Quick Reference

Commands

DDEDIT
Edits single-line text, dimension text, attribute definitions, and feature control frames.

PROPERTIES
Controls properties of existing objects.

TEXT
Creates a single-line text object.

System Variables

TEXTED
Specifies the user interface displayed for editing single-line text.

TEXTFILL
Controls the filling of TrueType fonts while printing.

TEXTQLTY
Sets the resolution tessellation fineness of text outlines.

Change Multiline Text

You can change the location and content of multiline text objects with the Properties Inspector palette, the In-Place Text Editor, and grips.

After you create multiline text, you can use the Properties Inspector palette to change

- Text style assignment
- Justification
- Width
- Rotation
- Line spacing
In addition, you can use the following to modify individual formatting, such as boldface and underlining, and width for multiline text objects:

- Text Editor visor
- In-Place Text Editor
- Grips

**Change Text Location**

You can use many of the common modifying commands and grips to move multiline text objects. A multiline text object has grips at the four corners of the text boundary and, in some cases, at the justification point.

Commands such as DIMLINEAR or LEADER create multiline text automatically without requiring that a bounding box be specified; these objects have only a single grip at the justification point.

When you need to align or move multiline text objects, you can use the Node and Insertion object snaps for precision. If the OSNAPNODELEGACY system variable is set to 0, the Node object snap ignores multiline text.

**See also:**

- [Work with Text Styles](#) on page 738
- [Control the Display of Polylines, Hatches, Gradient Fills, Lineweights, and Text](#) on page 192

**Quick Reference**

**Commands**

DDEDIT

Edits single-line text, dimension text, attribute definitions, and feature control frames.

FIND

Finds the text that you specify, and can optionally replace it with other text.

MTEDIT

Edits multiline text.
PROPERTIES

Controls properties of existing objects.

STYLE

Creates, modifies, or specifies text styles.

System Variables

CENTERMT

Controls how grips stretch multiline text that is centered horizontally.

MIRRTEXT

Controls how MIRROR reflects text.

OSNAPNODELEGACY

Controls whether the Node object snap can be used to snap to multiline text objects.

TEXTFILL

Controls the filling of TrueType fonts while printing.

TEXTEXTQLTY

Sets the resolution tessellation fineness of text outlines.

Find and Replace Text

You can easily find and replace text with the FIND command.

To search for and replace text, use FIND. Replacement is based on text content only; character formatting and text properties are not changed.

When searching for text in a 3D environment, the viewport will temporarily change to a 2D viewport so that text isn’t blocked by 3D objects in your drawing.

With FIND, you can use wild-card characters in your search.

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

**Check Spelling**

You can check the spelling of all text as it is entered in your drawing. You can also specify the specific language dictionary that is used and customize and manage multiple custom spelling dictionaries.

You can check the spelling of all text objects in your drawing, including

- Single and multilinete text
- Dimension text
- Multileader text
- Text within block attributes
- Text within xrefs

With Check Spelling, your drawing or the areas of your drawing’s text that you specify are searched for misspelled words. If a misspelled word is identified,
the word is highlighted and the drawing area zooms to that word in a scale that is easy to read.

**Check Spelling As You Type**

By default, you can check spelling as you enter text in the In-Place Text Editor. Any word you enter is checked for spelling errors when it is completed. A word is considered completed when one of the following actions are taken:

- Pressing Spacebar or Enter
- Moving the cursor to another position within the In-Place Text Editor.

**Misspelled words are underlined with a red dotted line**

Any word not found in the current dictionary is underlined as misspelled. Spelling suggestions are displayed when you right-click the underlined word.

**Quick Reference**

**Commands**

SPELL

Checks spelling in a drawing.
A table is a rectangular array of cells that contain annotation, primarily text but also blocks. Tables appear in many different forms on many of the sheets that make up drawing sets. In the AEC industry, tables are often referred to as “schedules” and contain information about the materials needed for the construction of the building being designed. In the manufacturing industry, they are often referred to as “BOM” (bills of materials).

The table object creates a table of any size that can be used for any purpose, including as a list or index to a set of drawing sheets.

Create and Modify Tables

A table is an object that contains data in rows and columns. A table object can be created from an empty table or table style.

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 Inspector or grips.
When you change the height or width of the table, only the row on page 1028 or column on page 1012 adjacent to the grip you have selected will change. The table will maintain its height or width. To change the size of the table proportionally to the size of the row or column you are editing, press Ctrl while using a column grip.

![Diagram of table changes](image)

**Break Tables into Multiple Parts**

A table with a large amount of data can be broken into primary and secondary table fragments. Use the table breaking grips found at the bottom of your table to make a table span multiple columns in your drawing or to manipulate the different table parts you have already created.

**Modify a Table Cell**

You can click inside a cell to select it. Grips are displayed in the middle of the cell borders. Click inside another cell to move selection to that cell. Drag the grips on a cell to make the cell and its column or row larger or smaller.

![Diagram of cell modifications](image)

**NOTE** When a cell is selected, double-click to edit the cell text. You can also start entering text when a cell is highlighted to replace its current content.
To select more than one cell, click and drag over several cells. You can also hold down Shift and click inside another cell to select those two cells and all the cells between them.

When you click inside a table cell, the Table Cell visor is displayed. From here, you can

- Insert and delete rows and columns
- Merge and unmerge cells
- Match cell styles
- Insert blocks, fields, and formulas
- Size rows and columns equally
- Remove all property overrides

**Customize Display of Column Letters and Row Numbers**

By default, the In-Place Text Editor displays column letters and row numbers when a table cell is selected for editing. Use the TABLEINDICATOR system variable to turn this display on and off.

**See also:**

- [Add Text and Blocks to Tables](#) on page 762

**Quick Reference**

**FIELD**

Creates a multiline text object with a field that can be updated automatically as the field value changes.

**FIND**

Finds the text that you specify, and can optionally replace it with other text.

**INSERT**

Inserts a block or drawing into the current drawing.

**LIST**

Displays property data for selected objects.
MTEXT
   Creates a multiline text object.
SPELL
   Checks spelling in a drawing.
TABLE
   Creates an empty table object.
TABLEEDIT
   Edits text in a table cell.
TABLEEXPORT
  Exports data from a table object in CSV file format.
UPDATEFIELD
   Manually updates fields in selected objects in the drawing.
TABLESTYLE
   Sets the name of the current table style.
FIELDDISPLAY
   Controls whether fields are displayed with a gray background.
FIELDVAL
   Controls how fields are updated.

Work with Table Styles

The appearance of the table is controlled by its table style. You can use the default table style, STANDARD, or a custom table style saved in the drawing.

**NOTE** AutoCAD 2011 for Mac does not support the ability to create or modify table and cell styles. You can edit the properties of a table and individual cells using the Properties Inspector.

When you select a table, you can change the appearance of the a table under the Table and Table Breaks sections of the Properties Inspector and Table visor. To display all of the table properties in the Properties Inspector, click All below the Object drop-down list. If you have a table style saved in your drawing, you can assign it to the table using the Table Style property under the Table section.
Table styles control the appearance of a table and all of the cells contained in
the table, but you can override the style of individual cells. The Cell and
Contents sections of the Properties Inspector are used to control the border
styles, text formatting, and the size of the cells.

The border properties in a table's cell style control the display of the gridlines
that divide the table into cells. The borders of the title row, the column heads
row, and the data rows can have different lineweight and color and can be
displayed or not displayed.

The appearance of text in the cells of the table is controlled by the text style
that is specified in the current cell style. You can use any text style in the
drawing or create a new one. The type of data you display in a row and the
formatting for that data type is controlled by the formatting options you select
in the Table Cell Format dialog box.

See also:
■ Work with Text Styles on page 738

Quick Reference

FIND
Finds the text that you specify, and can optionally replace it with other text.

INSERT
Inserts a block or drawing into the current drawing.

LIST
Displays property data for selected objects.

MTEXT
Creates a multiline text object.

SPELL
Checks spelling in a drawing.

TABLE
Creates an empty table object.

TABLEEDIT
Edits text in a table cell.
TABLEEXPORT
Exports data from a table object in CSV file format.

UPDATEFIELD
Manually updates fields in selected objects in the drawing.

CTABLESTYLE
Sets the name of the current table style.

FIELDDISPLAY
Controls whether fields are displayed with a gray background.

FIELDVAL
Controls how fields are updated.

Add Text and Blocks to Tables
Table cell data can include text and multiple blocks.

When a table is created, the first cell is highlighted, and you can begin entering text. The row height of the cell increases to accommodate the number of lines of text. To move to the next cell, press Tab, or use the arrow keys to move left, right, up, and down. You can quickly edit cell text by double-clicking in a selected cell or start entering text to replace the current content of a cell.

When you insert a block into a table cell, either the block can be automatically fit to the size of the cell, or the cell can adjust to accommodate the size of the block.

See also:
■ Use Fields in Text on page 732
■ Create Multiline Text on page 711

Quick Reference

FIELD
Creates a multiline text object with a field that can be updated automatically as the field value changes.
FIND
Finds the text that you specify, and can optionally replace it with other text.

INSERT
Inserts a block or drawing into the current drawing.

LIST
Displays property data for selected objects.

MATCHCELL
Applies the properties of a selected table cell to other table cells.

MTEXT
Creates a multilime text object.

SPELL
Checks spelling in a drawing.

TABLE
Creates an empty table object.

TABLEEDIT
Edits text in a table cell.

TABLEEXPORT
Exports data from a table object in CSV file format.

UPDATEFIELD
Manually updates fields in selected objects in the drawing.

CTABLESTYLE
Sets the name of the current table style.

FIELDDISPLAY
Controls whether fields are displayed with a gray background.

FIELDVALUE
Controls how fields are updated.
Use Formulas in Table Cells

Table cells can contain formulas that do calculations using the values in other table cells.

With a table cell selected, you can insert formulas from the Table Cell visor. You can also open the In-Place Text Editor and enter a formula in a table cell manually.

Insert a Formula

In formulas, cells are referred to by their column letter and row number. For example, the cell at top left in the table is A1. Merged cells use the number of what would be the top-left cell. A range of cells is defined by the first and last cells, with a colon between them. For example, the range A5:C10 includes cells in rows 5 through 10 in columns A, B, and C.

A formula must start with an equal sign (=). The formulas for sum, average, and count ignore empty cells and cells that do not resolve to a numeric value. Other formulas display an error (#) if any cell in the arithmetic expression is empty or contains nonnumeric data.

Use the Cell option to select a cell in another table in the same drawing. When you have selected the cell, the In-Place Text Editor opens so you can enter the rest of the formula.

Copy a Formula

When you copy a formula to another cell in the table, the range changes to reflect the new location. For example, if the formula in A10 sums A1 through A9, when you copy it to B10, the range of cells changes so that it sums B1 through B9.

If you don’t want a cell address to change when you copy and paste the formula, add a dollar sign ($) to the column or row part of the address. For example, if you enter $A10, the column stays the same and the row changes. If you enter $A$10, both column and row stay the same.

Insert Data Automatically

You can automatically increment data in adjacent cells within a table by using the AutoFill grip. For example, a table with a date column can have the dates automatically entered by entering the first necessary date and dragging the AutoFill grip.

Numbers will fill automatically by increments of 1 if one cell is selected and dragged. Similarly, dates will resolve by increments of one day if only one cell
is selected. If two cells are manually filled with dates one week apart, the remaining cells are incremented by one week.

**See also:**
- Use Fields in Text on page 732

**Quick Reference**

**FIELD**
  Creates a multiline text object with a field that can be updated automatically as the field value changes.

**MTEXT**
  Creates a multiline text object.

**TABLE**
  Creates an empty table object.

**TABLEEXPORT**
  Exports data from a table object in CSV file format.

**UPDATEFIELD**
  Manually updates fields in selected objects in the drawing.

**CTABLESTYLE**
  Sets the name of the current table style.

**FIELDDISPLAY**
  Controls whether fields are displayed with a gray background.

**FIELDDEVAL**
  Controls how fields are updated.

**TABLEINDICATOR**
  Controls the display of row numbers and column letters when the In-Place Text Editor is open for editing a table cell.
Dimensions and Tolerances

You can add measurements to your drawing with several dimensioning commands. Use dimension styles to format dimensions quickly and maintain industry or project dimensioning standards.

Understand Basic Concepts of Dimensioning

You can create several types of dimensions, and you can control their appearance by setting up dimension styles or by editing individual dimensions.

Overview of Dimensioning

Dimensioning is the process of adding measurement annotation to a drawing.

You can create dimensions for a variety of object types in many orientations. The basic types of dimensioning are

- Linear
- Radial (radius, diameter and jogged)
- Angular
- Ordinate
- Arc Length

Linear dimensions can be horizontal, vertical, aligned, rotated, baseline, or continued (chained). Some examples are shown in the illustration.
NOTE To simplify drawing organization and dimension scaling, it is recommended that you create dimensions on layouts rather than in model space.

Quick Reference

Commands

DIMANGULAR
    Creates an angular dimension.

DIMARC
    Creates an arc length dimension.

DIMBREAK
    Breaks or restores dimension and extension lines where they cross other objects.

DIMDIAMETER
    Creates a diameter dimension for a circle or an arc.

DIMEDIT
    Edits dimension text and extension lines.

DIMBREAK
    Adds or removes inspection information for a selected dimension.

DIMJOGGED
    Creates jogged dimensions for circles and arcs.
DIMBREAK
   Adds or removes a jog line on a linear or aligned dimension.
DIMLINEAR
   Creates a linear dimension.
DIMORDINATE
   Creates ordinate dimensions.
DIMRADIUS
   Creates a radius dimension for a circle or an arc.
DIMREASSOCIATE
   Associates or reassociates selected dimensions to objects or points on objects.
DIMBREAK
   Adjusts the spacing between linear dimensions or angular dimensions.
DIMSTYLE
   Creates and modifies dimension styles.
DIMTEDIT
   Moves and rotates dimension text and relocates the dimension line.
PROPERTIES
   Controls properties of existing objects.
QDIM
   Creates a series of dimensions quickly from selected objects.

System Variables
DIMASSOC
   Controls the associativity of dimension objects and whether dimensions are exploded.

Parts of a Dimension
Here is a list of the parts of a dimension along with their descriptions.
Dimensions have several distinct elements: dimension text, dimension lines, arrowheads, and extension lines.
Dimension text is a text string that usually indicates the measurement value. The text can also include prefixes, suffixes, and tolerances.

A dimension line indicates the direction and extent of a dimension. For angular dimensions, the dimension line is an arc.

Arrowheads, also called symbols of termination, are displayed at each end of the dimension line. You can specify different sizes and shapes for arrowheads or tick marks.

Extension lines, also called projection lines or witness lines, extend from the feature to the dimension line.

A center mark is a small cross that marks the center of a circle or arc.

Centerlines are broken lines that mark the center of a circle or arc.

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.
Associative Dimensions

Dimensions can be associative, nonassociative, or exploded. Associative dimensions adjust to changes in the geometric objects that they measure.

Dimension associativity defines the relationship between geometric objects and the dimensions that give their distance and angles. There are three types of associativity between geometric objects and dimensions.

- **Associative dimensions.** Automatically adjust their locations, orientations, and measurement values when the geometric objects associated with them are modified. Dimensions in a layout may be associated to objects in model space. The DIMASSOC system variable is set to 2.

- **Non-associative dimensions.** Selected and modified with the geometry they measure. Non-associative dimensions do not change when the geometric objects they measure are modified. The dimension variable DIMASSOC is set to 1.

- **Exploded dimensions.** Contain a collection of separate objects rather than a single dimension object. The DIMASSOC system variable is set to 0.

You can determine whether a dimension is associative or non-associative by selecting the dimension and doing one of the following:

- Use the Properties Inspector to display the properties of the dimension.
- Use the LIST command to display the properties of the dimension.

A dimension is considered associative even if only one end of the dimension is associated with a geometric object. The DIMREASSOCIATE command displays the associative and non-associative elements of a dimension.

**Special Situations and Limitations**

You may need to use DIMREGEN to update associative dimensions after panning or zooming, after opening a drawing that was modified with an earlier release, or after opening a drawing with external references that have been modified.

Although associative dimensions support most object types that you would expect to dimension, they do not support the following:

- Hatches
- Multiline objects
■ 2D solids
■ Objects with nonzero thickness
■ Images

When selecting objects to dimension, make sure that the objects that you select do not include a directly overlapping object that does not support associative dimensioning such as a 2D solid.

Associativity is not maintained between a dimension and a block reference if the block is redefined.

Associativity is not maintained between a dimension and a 3D solid if the shape of the 3D solid is modified.

Dimensions created with QDIM are not associative but may be associated individually with DIMREASSOCIATE.

For information about working with associative dimensions in combination with previous releases, see Save Drawings to Previous Drawing File Formats on page 931.

See also:
■ Change Dimension Associativity on page 835
■ Save Drawings to Previous Drawing File Formats on page 931

Quick Reference

Commands
DIMDISASSOCIATE
   Removes associativity from selected dimensions.
DIMREASSOCIATE
   Associates or reassociates selected dimensions to objects or points on objects.
DIMREGEN
   Updates the locations of all associative dimensions.
EXPLODE
   Breaks a compound object into its component objects.
LIST
Displays property data for selected objects.

OPTIONS
Customizes the program settings.

System Variables

DIMASSOC
Controls the associativity of dimension objects and whether dimensions are exploded.

Use Dimension Styles
You can control the appearance of dimensions by changing settings. For convenience and to help maintain dimensioning standards, you can store these settings in dimension styles.

Overview of Dimension Styles
A dimension style is a named collection of dimension settings that controls the appearance of dimensions, such as arrowhead style, text location, and lateral tolerances.

You create dimension styles to specify the format of dimensions quickly, and to ensure that dimensions conform to industry or project standards.

- When you create a dimension, it uses the settings of the current dimension style
- If you change a setting in a dimension style, all dimensions in a drawing that use the style update automatically
- You can create dimension substyles that, for specified types of dimensions, deviate from the current dimension style
- If necessary, you can override a dimension style temporarily
Quick Reference

Commands
DIMSTYLE
   Creates and modifies dimension styles.

Compare Dimension Styles and Variables
You can view all the settings in a dimension style. Dimension styles used in externally referenced drawings are differentiated from those defined in your current drawing.

You can list the dimension styles in the current drawing. You can also list all dimensioning system variables and their current status or only the variables affected by a dimension style.

When you list the current status of all dimensioning system variables, any running overrides that apply to the current dimension style are listed. You can also list the differences between a named dimension style and the current dimension style.

Use Externally Referenced Dimension Styles
The program displays externally referenced dimension style names using the same syntax as for other externally dependent named objects. When you view externally referenced dimension styles using the Dimension Style Manager, the name of the xref displays in the Styles list as Xref: ‘drawing name’ with each xref style appearing below the drawing name.

For example, if the drawing file baseplat.dwg has a dimension style called FRACTIONAL-1, and you attach baseplat.dwg as an xref to a new drawing, then the xref dimension style is displayed in the Styles list of the Dimension Style Manager as Xref: ‘baseplat.dwg’, and FRACTIONAL-1 appears under the drawing name.

Externally referenced dimension styles can be examined, but they cannot be modified or made current. You can use an externally referenced dimension style as a template for creating a new dimension style in your current drawing.
Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

Control Dimension Geometry

You can control the appearance of dimension lines, extension lines, arrowheads, and center marks.

Control Dimension Lines

You can control dimension line properties including color, lineweight, and spacing.

You can control several aspects of a dimension line. You can

■ Specify color and lineweight for visual effect and printing

■ 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.

Quick Reference

**Commands**

**DIMSTYLE**

Creates and modifies dimension styles.

**System Variables**

**DIMCLR**

Assigns colors to dimension lines, arrowheads, and dimension leader lines.

**DIMDLE**

Sets the distance the dimension line extends beyond the extension line when oblique strokes are drawn instead of arrowheads.

**DIMDLI**

Controls the spacing of the dimension lines in baseline dimensions.

**DIMGAP**

Sets the distance around the dimension text when the dimension line breaks to accommodate dimension text.

**DIMLT**

Sets the linetype of the dimension line.

**DIMLWD**

Assigns lineweight to dimension lines.

**DIMSD1**

Controls suppression of the first dimension line and arrowhead.
DIMSD2
Controls suppression of the second dimension line and arrowhead.

DIMSOXD
Suppresses arrowheads if not enough space is available inside the extension lines.

DIMTOFL
Controls whether a dimension line is drawn between the extension lines even when the text is placed outside.

Control Extension Lines
You can control extension line properties including color, lineweight, overshoot, and offset length.

You can
- Specify color and lineweight for visual effect and printing
- Suppress one or both extension lines if they are unnecessary, or if there is not enough space
- Specify how far beyond from the dimension line the extension line extends (overshoot)
- Control the extension origin offset, the distance between the extension line origin, and the start of the extension line
Specify a fixed length for extension lines, as measured from the dimension line toward the extension line origin.

Specify a noncontinuous linetype, typically used for centerlines.

Modify the angle of the extension lines of a selected dimension to make them oblique.

Fixed-Length Extension Lines
You can specify a dimension style that sets the total length for extension lines starting from the dimension line toward the dimension origin point.
The extension line offset distance from the origin will never be less than the value specified by the DIMEXO system variable.

See also:
- Create Dimensions with Oblique Extension Lines on page 808

Quick Reference

Commands
DIMSTYLE
Creates and modifies dimension styles.

System Variables
DIMCLRE
Assigns colors to extension lines, center marks, and centerlines.
DIMDLE
Sets the distance the dimension line extends beyond the extension line when oblique strokes are drawn instead of arrowheads.
DIMEXE
Specifies how far to extend the extension line beyond the dimension line.
DIMEXO
Specifies how far extension lines are offset from origin points.
DIMFXL
Sets the total length of the extension lines starting from the dimension line toward the dimension origin.
DIMFXLON
Controls whether extension lines are set to a fixed length.
DIMLTEX1
Sets the linetype of the first extension line.
DIMLTEX2
Sets the linetype of the second extension line.
DIMLWE
Assigns linewidth to extension lines.

DIMSE1
Suppresses display of the first extension line.

DIMSE2
Suppresses display of the second extension line.

**Control Dimension Arrowheads**

You can control the arrowhead symbols in dimensions and leaders including their type, size, and visibility.

You can choose from many standard types of arrowheads, or you can create your own arrowheads. Additionally, you can

- Suppress the display of arrowheads, or use one arrowhead only
- Apply a different type of arrowhead to each end of a dimension line
- Control the size of arrowheads
- Flip the direction of an arrowhead using the dimension shortcut menu

**NOTE** Flipped arrowheads maintain their appearance in versions later than AutoCAD 2002. However, if you edit a drawing with flipped arrowheads in a release earlier than AutoCAD 2006, the arrowhead directions will revert to their original orientations.

See also:
- Customize Arrowheads on page 781

**Quick Reference**

**Commands**

DIMSTYLE
Creates and modifies dimension styles.
System Variables

DIMCLRD
Assigns colors to dimension lines, arrowheads, and dimension leader lines.

DIMDLE
Sets the distance the dimension line extends beyond the extension line when oblique strokes are drawn instead of arrowheads.

DIMSD1
Controls suppression of the first dimension line and arrowhead.

DIMSD2
Controls suppression of the second dimension line and arrowhead.

Customize Arrowheads

You can create your own custom arrowheads.

Arrowheads are stored as block definitions. To use your own arrowhead, provide the name of an existing block definition. For information about creating blocks, see Create Blocks Within a Drawing on page 413.

NOTE Annotative blocks cannot be used as custom arrowheads for dimensions or leaders.

Arrowhead sizing relies on the overall dimension scale factor. When you create a dimension, the block is inserted where the arrowheads would normally go. The object’s X and Y scale factors are set to arrowhead size overall scale. The dimension line is trimmed by text gap x overall scale units at each end. To trim the dimension line, the rightmost block is inserted with a zero rotation angle for horizontal dimensioning. The leftmost block is rotated 180 degrees about its insertion point.

NOTE The insertion point a block is defined with affects its placement as a custom arrowhead on a dimension or leader. For information on changing the insertion point of a block, see Create Drawing Files for Use as Blocks on page 414.

If you use paper-space scaling, the scale factor is computed before applying it to the arrowhead size value.

See also:

- Create Blocks Within a Drawing on page 413
Create Drawing Files for Use as Blocks on page 414

Quick Reference

Commands

BLOCK
Creates a block definition from selected objects.

DIMSTYLE
Creates and modifies dimension styles.

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

System Variables

DIMASZ
Controls the size of dimension line and leader line arrowheads.

DIMBLK
Sets the arrowhead block displayed at the ends of dimension lines.

DIMBLK1
Sets the arrowhead for the first end of the dimension line when DIMSAH is on.

DIMBLK2
Sets the arrowhead for the second end of the dimension line when DIMSAH is on.

DIMCLRDA
Assigns colors to dimension lines, arrowheads, and dimension leader lines.

DIMDLE
Sets the distance the dimension line extends beyond the extension line when oblique strokes are drawn instead of arrowheads.

DIMSAH
Controls the display of dimension line arrowhead blocks.
DIMTSZ

Specifies the size of oblique strokes drawn instead of arrowheads for linear, radius, and diameter dimensioning.

Control Dimension Text

You can control the placement of dimension text, arrowheads, and leader lines relative to the dimension and extension lines.

Fit Dimension Text Within Extension Lines

Dimension text and arrowheads usually appear between the extension lines when there is enough space. You can specify how these elements are placed when space is limited.

Many factors, such as the size of extension line spacing and arrowhead size, influence how dimension text and arrowheads fit within the extension lines. In general, the best fit, given the available space, is applied. If possible, both text and arrowheads are accommodated between the extension lines, no matter what fit option you choose.

When creating new dimensions, you can choose to place text by entering a coordinate or using the pointing device; this is known as user-defined text placement. Alternatively, the program can compute the text position for you. The options for automatic fitting of text and arrowheads are listed in the Modify/New Dimension Style dialog box, Fit tab.

For example, you can specify that text and arrowheads be kept together. In this case, if there is not room for both between the extension lines, they are both placed outside. You can specify that if there is room for only text or arrowheads, then either text only or arrowheads only are placed between the extension lines.

The following illustrations show how the program applies a "best fit" for arrowheads and text.
If there is no room for text between the extension lines, you can have a leader line created automatically. This is useful in cases where text outside the extension lines would interfere with other geometry, for example, in continued dimensions. Whether text is drawn to the right or the left of the leader is controlled by the horizontal justification setting in the Modify/New Dimension Style dialog box, Text tab. Also, you can fit text and arrowheads by changing their size.

Even if the arrowheads are outside the extension lines, you can have a line drawn between the extension lines. This is called forcing an internal line and is illustrated as follows.

**Fit Diameter Dimension Text**

You can draw several different diameter dimensions depending on text placement, horizontal settings on the Modify/New Dimension Style dialog box, Text tab, and whether you select the Draw Dim Line Between Ext Lines option on the Modify/New Dimension Style dialog box, Fit tab.
Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

System Variables

DIMATFIT

Determines how dimension text and arrows are arranged when space is not sufficient to place both within the extension lines.

DIMJUST

Controls the horizontal positioning of dimension text.

DIMLWD

Assigns lineweight to dimension lines.

DIMTAD

Controls the vertical position of text in relation to the dimension line.

DIMTXTDIRECTION

Specifies the reading direction of the dimension text.

DIMTHH

Controls the position of dimension text inside the extension lines for all dimension types except Ordinate.
DIMTIX
Draws text between extension lines.

DIMTOFL
Controls whether a dimension line is drawn between the extension lines even when the text is placed outside.

DIMTOH
Controls the position of dimension text outside the extension lines.

DIMTVP
Controls the vertical position of dimension text above or below the dimension line.

DIMUPT
Controls options for user-positioned text.

Control the Location of Dimension Text
You can locate dimension text manually and specify its alignment and orientation.

The program comes with several justification settings that facilitate compliance with international standards, or you can choose your own location for the text.

Many of the settings are interdependent. Example images in the Dimension Style Manager are updated dynamically to illustrate how text appears as you change the settings.
Align Dimension Text

Whether text is inside or outside the extension lines, you can choose whether it is aligned with the dimension line or remains horizontal. The following examples show two combinations of these options.

The default alignment is horizontal dimension text, even for vertical dimensions.

Position Dimension Text Horizontally

The position of the text along the dimension line in relation to the extension lines is referred to as text placement. To place text yourself when you create a dimension, use the Place Text Manually option on the Modify/New Dimension Style dialog box, Fit tab. Use the text placement options to automatically place text at the center of the dimension line, at either extension line, or over either extension line.

First and second extension lines are defined by the order in which you specified the extension line origins when you created the dimension. For angular
dimensions, the second extension line is counterclockwise from the first. In the following illustrations, 1 is the first extension line origin and 2 the second.

If you place text manually, you can place the dimension text anywhere along the dimension line, inside or outside the extension lines, as you create the dimension. This option provides flexibility and is especially useful when space is limited. However, the horizontal alignment options provide better accuracy and consistency between dimensions.

**Position Dimension Text Vertically**

The position of the text relative to the dimension line is referred to as vertical text placement. Text can be placed above or below or centered within the dimension line. In the ANSI standards, centered text usually splits the dimension line. In the ISO standards, it is usually above or outside the dimension line. For example, ISO standards permit angular dimension text to appear in any of the ways shown.

Other settings, such as Text Alignment, affect the vertical alignment of text. For example, if Horizontal Alignment is selected, text inside the extension
lines and centered within the dimension line is horizontal, as shown in the
leftmost illustration above. The text is horizontal even if the dimension line
is not itself horizontal.

Quick Reference

Commands
DIMSTYLE
Creates and modifies dimension styles.
DIMTEDIT
Moves and rotates dimension text and relocates the dimension line.

System Variables
DIMJUST
Controls the horizontal positioning of dimension text.
DIMTAD
Controls the vertical position of text in relation to the dimension line.
DIMTXTDIRECTION
Specifies the reading direction of the dimension text.
DIMTIH
Controls the position of dimension text inside the extension lines for all
dimension types except Ordinate.
DIMTOH
Controls the position of dimension text outside the extension lines.
DIMTPV
Controls the vertical position of dimension text above or below the dimension
line.
DIMUPT
Controls options for user-positioned text.
Control the Appearance of Dimension Text

You can include prefixes, suffixes, and user-supplied text in dimensions. You can also control the text style and formatting used in dimension text.

The program supports a mixture of user-supplied text, prefixes and suffixes supplied by the dimension style, and generated measurements. For example, you could add a diameter symbol as a prefix to a measurement or add the abbreviation for a unit, such as mm, as a suffix. Text in this context refers to all dimension text, prefixes and suffixes, primary and alternate units, and lateral tolerances. Geometric tolerances are controlled independently.

Dimension text is treated as a single string of text, which you create and format using your text editor.

Control the Text Style in Dimensions

The appearance of dimension text is governed by the text style selected in the Modify/New Dimension Style dialog box, Text tab. You can choose a text style while creating a dimension style and specify a text color and a height independent of the current text style's height setting. You can also specify the gap between base dimension text and the box that surrounds it.

The text styles used for dimensions are the same text styles used by all text created in your drawing.

For more information, see Work with Text Styles on page 738.

Supply User Text to Dimensions

In addition to the prefixes and suffixes specified for primary and alternate units, you can supply your own text as you create a dimension. Because the prefix, suffix, and user-supplied text form a single text string, you can represent tolerance stacks and apply changes to font, text size, and other characteristics using the text editor.

To add user text above and below the dimension line, use the separator symbol \X. Text that precedes this symbol is aligned with and above the dimension line. Text that follows the \X symbol is aligned with and below the dimension line. The space between the dimension line and the text is determined by the value you enter in Offset from Dim Lim on the Modify/New Dimension Style dialog box, Text tab.
Example: User Text in Dimensions

In this example, the primary dimension measurement is 5.08, and the alternate dimension measurement is 2.00. The primary units have the suffix H7/h6, and the alternate units have the suffix inches.

At the text prompt, while creating the dimension, you enter the following format string:

\(<> \text{H7/h6} \backslash X \text{See Note 26} \backslash P [ ]\)

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:

![Dimension Text Example]

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

System Variables

DIMCLRT

Assigns colors to dimension text.

DIMGAP

Sets the distance around the dimension text when the dimension line breaks to accommodate dimension text.

DIMTFAC

Specifies a scale factor for the text height of fractions and tolerance values relative to the dimension text height, as set by DIMTXT.

DIMTFILL

Controls the background of dimension text.
DIMTFILLCLR
Sets the color for the text background in dimensions.

DIMTXSTY
Specifies the text style of the dimension.

DIMTXT
Specifies the height of dimension text, unless the current text style has a fixed height.

DIMTXTDIRECTION
Specifies the reading direction of the dimension text.

Control Dimension Values
The numeric values displayed in dimensions can appear in several formats. You can also control how numeric distances are represented.

Control the Display of Dimension Units
The numeric values of dimensions can be displayed as a single measurement or in two measurement systems. In either case, you can control details of how the numeric values are presented.

The settings for primary units control the display of the dimension values, including the unit format, the numeric precision, and the decimal separator style. For example, you can enter the diameter symbol as a prefix, as shown in the illustration. Any prefix you specify replaces the prefixes normally used for diameter and radius dimensions (unicode 2205 and R, respectively).
These settings are available on the Modify/New Dimension Style dialog box, Primary Units tab.

**Control the Display of Alternate Units**

You can create dimensions in two systems of measurement simultaneously. A common use of this feature is to add feet and inches dimensions to drawings created using metric units. The alternate units appear in square brackets ( [ ] ) in the dimension text. Alternate units cannot be applied to angular dimensions.

If alternate-units dimensioning is on when you edit a linear dimension, the measurement is multiplied by an alternate scale value that you specify. This value represents the number of alternate units per current unit of measurement. The default value for imperial units is 25.4, which is the number of millimeters per inch. The default value for metric units is about 0.0394, which is the number of inches per millimeter. The number of decimal places is specified by the precision value for alternate units.

For example, for imperial units, if the alternate scale setting is the default value, 25.4, and the alternate precision is 0.00, the dimension might look like the following figure.

![Dimension Example](image)

**Quick Reference**

**Commands**

**DIMSTYLE**

Creates and modifies dimension styles.

**System Variables**

**DIMALT**

Controls the display of alternate units in dimensions.
DIMALTD  
Controls the number of decimal places in alternate units.

DIMALTF  
Controls the multiplier for alternate units.

DIMALTTD  
Sets the number of decimal places for the tolerance values in the alternate units of a dimension.

DIMALTU  
Sets the units format for alternate units of all dimension substiles except Angular.

DIMALTZ  
Controls the suppression of zeros for alternate unit dimension values.

DIMAPOST  
Specifies a text prefix or suffix (or both) to the alternate dimension measurement for all types of dimensions except angular.

DIMAUNIT  
Sets the units format for angular dimensions.

DIMDEC  
Sets the number of decimal places displayed for the primary units of a dimension.

DIMDSEP  
Specifies a single-character decimal separator to use when creating dimensions whose unit format is decimal.

DIMLFAC  
Sets a scale factor for linear dimension measurements.

DIMLUNIT  
Sets units for all dimension types except Angular.

DIMPOST  
Specifies a text prefix or suffix (or both) to the dimension measurement.
DIMTDEC
Sets the number of decimal places to display in tolerance values for the primary units in a dimension.

Round Off Dimension Values
You can round off the numeric values in dimensions and lateral tolerances.
You can round off all dimension values except those for angular dimensions. For example, if you specify a round-off value of 0.25, all distances are rounded to the nearest 0.25 unit. The number of digits displayed after the decimal point depends on the precision set for primary and alternate units and lateral tolerance values.

Quick Reference

Commands
DIMSTYLE
Creates and modifies dimension styles.

System Variables
DIMRND
Rounds all dimensioning distances to the specified value.

Suppress Zeros in Dimensions
You can suppress leading and trailing zeros in the numeric portion of dimension text. You can also specify the sub unit for the dimension distance.
If you suppress leading zeros in decimal dimensions, 0.500 becomes .500. If you suppress trailing zeros, 0.500 becomes .5. You can suppress both leading and trailing zeros so that 0.5000 becomes .5 and 0.0000 becomes 0.

For dimension distances less than one unit, you can set the dimension distance to display in sub units. If the distance is shown in *m*, you can set to display distances less than one *m* in *cm* or *mm*.

The table shows the effect of selecting each option and provides examples of the architectural units style. If feet are included with a fractional inch, the number of inches is indicated as zero, no matter which option you select. Thus, the dimension 4'-3/4" becomes 4'-0 3/4".

### Zero suppression for feet and inches

<table>
<thead>
<tr>
<th>Option</th>
<th>Effect</th>
<th>Examples</th>
</tr>
</thead>
<tbody>
<tr>
<td>No options selected</td>
<td>Includes zero feet and zero inches</td>
<td>0'-0 1/2&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0'-6&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1'-0&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3/4&quot;</td>
</tr>
<tr>
<td>0 Inches selected</td>
<td>Suppresses zero inches (includes zero feet)</td>
<td>0'-0 1/2&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0'-6&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1'</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1'-0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3/4&quot;</td>
</tr>
<tr>
<td>0 Feet selected</td>
<td>Suppresses zero feet (includes zero inches)</td>
<td>1/2&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>6&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1'-0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3/4&quot;</td>
</tr>
<tr>
<td>0 Feet and 0 Inches selected</td>
<td>Suppresses zero feet and zero inches</td>
<td>1/2&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>6&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1'</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1'-0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3/4&quot;</td>
</tr>
</tbody>
</table>

### Quick Reference

**Commands**

**DIMSTYLE**

Creates and modifies dimension styles.

**System Variables**

**DIMALTZ**

Controls suppression of zeros in tolerance values.

**DIMALTZ**

Controls the suppression of zeros for alternate unit dimension values.
DIMAZIN
Suppresses zeros for angular dimensions.

DIMTZIN
Controls the suppression of zeros in tolerance values.

DIMZIN
Controls the suppression of zeros in the primary unit value.

Display Lateral Tolerances

Lateral tolerances are values indicating the amount a measured distance can vary. You can control whether lateral tolerances are displayed and you can choose from several styles of lateral tolerances.

A lateral tolerance specifies the amount by which a dimension can vary. By specifying tolerances in manufacturing, you can control the degree of accuracy needed for a feature. A feature is some aspect of a part, such as a point, line, axis, or surface.

You can apply tolerances directly to a dimension by appending the tolerances to the dimension text. These dimension tolerances indicate the largest and smallest permissible size of the dimension. You can also apply geometric tolerances, which indicate deviations of form, profile, orientation, location, and runout.

Lateral tolerances can be specified from theoretically exact measurements. These are called basic dimensions and have a box drawn around them.

If the dimension value can vary in both directions, the plus and minus values you supply are appended to the dimension value as deviation tolerances. If the deviation tolerance values are equal, they are displayed with a sign and they are known as symmetrical. Otherwise, the plus value goes above the minus value.
If the tolerances are applied as limits, the program uses the plus and minus values you supply to calculate a maximum and minimum value. These values replace the dimension value. If you specify limits, the upper limit goes above the lower.

**Format Lateral Tolerances**

You can control the vertical placement of tolerance values relative to the main dimension text. Tolerances can align with the top, middle, or bottom of the dimension text.

Along with vertical placement of tolerance values, you can also control the horizontal alignment of the upper and lower tolerance values. The upper and lower tolerance values can be aligned using either the operational symbols or decimal separators.

You can also control zero suppression as you can with the primary and alternate units. Suppressing zeros in lateral tolerances has the same effect as...
suppressing them in the primary and alternate units. If you suppress leading zeros, 0.5 becomes .5, and if you suppress trailing zeros, 0.5000 becomes 0.5.

See also:
- Add Geometric Tolerances on page 837

Quick Reference

Commands

DIMSTYLE

Creates and modifies dimension styles.

System Variables

DIMALTSD

Sets the number of decimal places for the tolerance values in the alternate units of a dimension.

DIMALTTZ

Controls suppression of zeros in tolerance values.

DIMGAP

Sets the distance around the dimension text when the dimension line breaks to accommodate dimension text.

DIMLIM

Generates dimension limits as the default text.

DIMTDEC

Sets the number of decimal places to display in tolerance values for the primary units in a dimension.

DIMTFAC

Specifies a scale factor for the text height of fractions and tolerance values relative to the dimension text height, as set by DIMTXT.

DIMTM

Sets the minimum (or lower) tolerance limit for dimension text when DIMTOL or DIMLIM is on.
DIMTOL
Appends tolerances to dimension text.

DIMTP
Sets the maximum (or upper) tolerance limit for dimension text when DIMTOL or DIMLIM is on.

DIMTZIN
Controls the suppression of zeros in tolerance values.

Control the Display of Fractions
You can control the format of the fraction displayed in dimensions.

You can set the fraction format in dimensions using the DIMFRAC system variable when the DIMLUNIT system variable is set to 4 (architecture) or 5 (fractional).

The following illustration shows the different fraction formats available.

These settings are available on the Modify/New Dimension Style dialog box, Primary Units tab.

Quick Reference

Commands
DIMSTYLE
Creates and modifies dimension styles.

System Variables
DIMFRAC
Sets the fraction format when DIMLUNIT is set to 4 (Architectural) or 5 (Fractional).
DIMLUNIT

Sets units for all dimension types except Angular.

Set the Scale for Dimensions

You can specify the size of dimensions in your drawing. How you set dimension size depends on the method you use to lay out and print 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 printed size. Dimension scale does not apply the overall scale factor to tolerances or measured lengths, coordinates, or angles.

NOTE You can use annotative scaling to control the overall scale of dimensions displayed in layout viewports. When you create annotative dimensions, they are scaled based on the current annotation scale setting and automatically displayed at the correct size.

Setting dimension scale depends on how you lay out your drawing. There are three methods used to create dimensions in a drawing layout:

- **Dimension in model space for printing in model space**. This is the traditional method used with single-view drawings. To create dimensions that are scaled correctly for printing, set the DIMSCALE system variable to the inverse of the intended print scale. For example, if the print scale is 1/4, set DIMSCALE to 4.

- **Dimension in model space for printing in paper space**. This was the preferred method for complex, multiple-view drawings prior to AutoCAD 2002. Use this method when the dimensions in a drawing need to be referenced by other drawings (xrefs) or when creating isometric dimensions in 3D isometric views. To prevent the dimensions in one layout viewport from being displayed in other layout viewports, create a dimensioning layer for each layout viewport that is frozen in all other layout viewports. To create dimensions that are scaled automatically for display in a paper space layout, set the DIMSCALE system variable to 0.

- **Dimension in layouts**. This is the simplest dimensioning method. Dimensions are created in paper space by selecting model space objects or by specifying object snap locations on model space objects. By default, associativity between paper space dimensions and model space objects is
maintained. No additional scaling is required for dimensions created in a paper space layout: DIMLFAC and DIMSCALE do not need to be changed from their default value of 1.0000.

NOTE When you dimension model space objects in paper space using associative dimensions, dimension values for the display scale of each viewport are automatically adjusted. This adjustment is combined with the current setting for DIMLFAC and is reported by the LIST command as a dimension style override. For nonassociative dimensions, you must set DIMLFAC manually.

See also:

- Draw, Scale, and Annotate in Model Space on page 131
- Scale Views in Layout Viewports on page 147
- Scale Annotations on page 656

Quick Reference

Commands

DIMREGEN
Updates the locations of all associative dimensions.

DIMSTYLE
Creates and modifies dimension styles.

System Variables

DIMASSOC
Controls the associativity of dimension objects and whether dimensions are exploded.

DIMLFAC
Sets a scale factor for linear dimension measurements.

DIMSCALE
Sets the overall scale factor applied to dimensioning variables that specify sizes, distances, or offsets.
Create Dimensions
You can create all of the standard types of dimensions.

Create Linear Dimensions
You can create linear dimensions with horizontal, vertical, and aligned dimension lines. These linear dimensions can also be stacked, or they can be created end to end.

Overview of Creating Linear Dimensions
Linear dimensions can be horizontal, vertical, or aligned. With aligned dimensions, the dimension line is parallel to the line (imaginary or real) between the extension line origins. Baseline (or parallel) and continued (or chain) dimensions are series of consecutive dimensions that are based on a linear dimension.

In all four illustrations, the extension line origins are designated explicitly at 1 and 2, respectively. The dimension line location is specified at 3.

As you create linear dimensions, you can modify the content of the text, the angle of the text, or the angle of the dimension line.
Quick Reference

Commands

DIMALIGNED
Creates an aligned linear dimension.

DIMBASELINE
Creates a linear, angular, or ordinate dimension from the baseline of the previous or selected dimension.

DIMCONTINUE
Creates a dimension that starts from an extension line of a previously created dimension.

DIMEDIT
Edits dimension text and extension lines.

DIMLINEAR
Creates a linear dimension.

DIMSTYLE
Creates and modifies dimension styles.

QDIM
Creates a series of dimensions quickly from selected objects.

System Variables

DIMDLI
Controls the spacing of the dimension lines in baseline dimensions.

Create Horizontal and Vertical Dimensions

You can create dimensions using only the horizontal or vertical components of the locations or objects that you specify.

The program automatically applies a horizontal or vertical dimension according to the extension line origins that you specify or the location where you select an object; however, you can override this as you create the dimension by specifying that a dimension be horizontal or vertical. For example, in the
following illustration, a horizontal dimension is drawn by default unless you specify a vertical one.

![Horizontal and Vertical Dimensions](image)

**Quick Reference**

**Commands**

DIMLINEAR

Creates a linear dimension.

**System Variables**

DIMEXO

Specifies how far extension lines are offset from origin points.

**Create Aligned Dimensions**

You can create dimensions that are parallel to the locations or objects that you specify.

In aligned dimensions, the dimension line is parallel to the extension line origins. The illustration shows two examples of aligned dimensioning. The object is selected (1), and the location of the aligned dimension is specified (2). The extension lines are drawn automatically.
Quick Reference

Commands
DIMALIGNED
Creates an aligned linear dimension.

DIMSTYLE
Creates and modifies dimension styles.

System Variables
DIMEXO
Specifies how far extension lines are offset from origin points.

Create Baseline and Continued Dimensions
Baseline dimensions are multiple dimensions measured from the same baseline. Continued dimensions are multiple dimensions placed end to end.
You must create a linear, aligned, or angular dimension before you create baseline or continued dimensions. You create baseline dimensions incrementally from the most recently created dimension in the current session.

Both baseline and continued dimensions are measured from the previous extension line unless you specify another point as the point of origin.
Quick Reference

Commands

DIMBASELINE
Creates a linear, angular, or ordinate dimension from the baseline of the previous or selected dimension.

DIMCONTINUE
Creates a dimension that starts from an extension line of a previously created dimension.

DIMSTYLE
Creates and modifies dimension styles.

System Variables

DIMDLI
Controls the spacing of the dimension lines in baseline dimensions.

Create Rotated Dimensions

In rotated dimensions, the dimension line is placed at an angle to the extension line origins.

The illustration shows an example of a rotated dimension. In the example, the angle specified for dimension rotation is equal to the angle of the slot.
Quick Reference

Commands

DIMALIGNED
Creates an aligned linear dimension.

DIMLINEAR
Creates a linear dimension.

Create Dimensions with Oblique Extension Lines

You can create dimensions with extension lines that are not perpendicular to their dimension lines.

Extension lines are created perpendicular to the dimension line. However, if the extension lines conflict with other objects in a drawing, you can change their angle after the dimension has been drawn.

New dimensions are not affected when you make an existing dimension oblique.

808 | Chapter 29  Dimensions and Tolerances
Quick Reference

Commands

DIMEDIT

Edits dimension text and extension lines.

Create Radial Dimensions

Radial dimensions measure the radii and diameters of arcs and circles with optional centerlines or a center mark.

There are two types of radial dimensions:

■ DIMRADIUS measures the radius of an arc or circle, and displays the dimension text with the letter \( R \) in front of it.

■ DIMDIAMETER measures the diameter of an arc or circle, and displays the dimension text with the diameter symbol in front of it.

For horizontal dimension text, if the angle of the radial dimension line is greater than 15 degrees from horizontal, a hook line, also called a dogleg or landing, one arrowhead long, is created next to the dimension text.

Control Extension Lines

When an arc is dimensioned, the radial or diametric dimension does not have to be positioned along the arc directly. If a dimension is positioned past the end of an arc, either an extension line will be drawn that follows the path of the arc being dimensioned or no extension line will be drawn. When the extension line is suppressed (off), the dimension line of the radial or diametric
dimension is drawn through the center point of the arc instead of to the extension line.

The DIMSE1 system variable controls whether or not a radial or diametric dimension will be drawn with an extension line when it is positioned off the end of an arc. When the display of the arc extension line is not suppressed, a gap between the arc and arc extension line is made. The size of the gap drawn is controlled with the DIMEXO system variable.

**Control Centerlines and Center Marks**

Depending on your dimension style settings, center marks and lines generate automatically for diameter and radius dimensions. They are created only if the dimension line is placed outside the circle or arc. You can create centerlines and center marks directly with the DIMCENTER command.

You can control the size and visibility of centerlines and center marks on the New/Modify Dimension Style dialog box, Symbols and Arrows tab, under Center Marks. You can also access this setting with the DIMCEN system variable.
The size of the centerline is the length of the centerline segment that extends outside the circle or arc. It is also the size of the gap between the center mark and the start of the centerline. The size of the center mark is the distance from the center of the circle or arc to the end of the center mark.

Create Jogged Radius Dimensions

With the DIMJOGGED command, you can create jogged radius dimensions, also called “foreshortened radius dimensions,” when the center of an arc or circle is located off the layout and cannot be displayed in its true location. The origin point of the dimension can be specified at a more convenient location called the center location override.

You can control the default angle of the jog in the New/Modify Dimension Style dialog box, Symbols and Arrows tab, under Radius Dimension Jog.
Once a jogged radius dimension is created, you can modify the jog and the center location override by

- Using grips to move the features
- Changing the locations of the features with the Properties Inspector
- Using STRETCH

**NOTE** Jogged radius dimensions can be viewed but not edited in versions previous to AutoCAD 2006. Also, if you make dramatic changes to the associated geometry, you may get unpredictable results for the jogged radius dimension.

See also:

- [Fit Dimension Text Within Extension Lines](#) on page 783

**Quick Reference**

**Commands**

**DIMCENTER**

Creates the center mark or the centerlines of circles and arcs.

**DIMDIAMETER**

Creates a diameter dimension for a circle or an arc.

**DIMJOGGED**

Creates jogged dimensions for circles and arcs.

**DIMRADIUS**

Creates a radius dimension for a circle or an arc.
DIMSTYLE
Stores the name of the current dimension style.

QDIM
Creates a series of dimensions quickly from selected objects.

**System Variables**

DIMATFIT
Determines how dimension text and arrows are arranged when space is not sufficient to place both within the extension lines.

DIMCEN
Controls drawing of circle or arc center marks and centerlines by the DIMCENTER, DIMDIAMETER, and DIMRADIUS commands.

DIMEXO
Specifies how far extension lines are offset from origin points.

DIMJOGANG
Determines the angle of the transverse segment of the dimension line in a jogged radius dimension.

DIMJUST
Controls the horizontal positioning of dimension text.

DIMSE1
 Suppresses display of the first extension line.

DIMTAD
Controls the vertical position of text in relation to the dimension line.

DIMTXTDIRECTION
Specifies the reading direction of the dimension text.

DIMTIH
Controls the position of dimension text inside the extension lines for all dimension types except Ordinate.

DIMTMOVE
Sets dimension text movement rules.
**DIMTOFL**

Controls whether a dimension line is drawn between the extension lines even when the text is placed outside.

**DIMTOH**

Controls the position of dimension text outside the extension lines.

**DIMUPT**

Controls options for user-positioned text.

---

**Create Angular Dimensions**

Angular dimensions measure the angle between two lines or three points.

To measure the angle between two radii of a circle, you select the circle and specify the angle endpoints. With other objects, you select the objects and then specify the dimension location. You can also dimension an angle by specifying the angle vertex and endpoints. As you create the dimension, you can modify the text content and alignment before specifying the dimension line location.

**NOTE** You can create baseline and continued angular dimensions relative to existing angular dimensions. Baseline and continued angular dimensions are limited to 180 degrees or less. To obtain baseline and continued angular dimensions larger than 180 degrees, use grip editing to stretch the location of the extension line of an existing baseline or continued dimension.

---

**Dimension Lines**

If you use two straight, nonparallel lines to specify an angle, the dimension line arc spans the angle between the two lines. If the dimension line arc does not meet one or both of the lines being dimensioned, the program draws one or two extension lines to intersect the dimension line arc. The arc is always less than 180 degrees.

**Dimension Circles and Arcs**

If you use an arc or a circle or three points to specify an angle, the program draws the dimension line arc between the extension lines. The extension lines are drawn from the angle endpoints to the intersection of the dimension line arc.
The location that you specify for the dimension line arc determines the quadrant of the dimensioned angle.

**Dimension to a Quadrant**

Angular dimensions can measure a specific quadrant that is formed when dimensioning the angle between of the endpoints of a line or arc, center point of a circle, or two vertices. As an angular dimension is being created, there are four possible angles that can be measured. By specifying a quadrant it allows you to ensure that the correct angle is dimensioned. When placing an angular dimension after a quadrant has been specified, you can place the dimension text outside of the extension lines of the dimension. The dimension line is automatically extended.

**Quick Reference**

**Commands**

**DIMANGULAR**

Creates an angular dimension.

**DIMBASELINE**

Creates a linear, angular, or ordinate dimension from the baseline of the previous or selected dimension.
DIMCONTINUE

Creates a dimension that starts from an extension line of a previously created dimension.

System Variables

DIMADEC

Controls the number of precision places displayed in angular dimensions.

DIMAUNIT

Sets the units format for angular dimensions.

DIMDEC

Sets the number of decimal places displayed for the primary units of a dimension.

Create Ordinate Dimensions

Ordinate dimensions measure the perpendicular distance from an origin point called the datum to a feature, such as a hole in a part. These dimensions prevent escalating errors by maintaining accurate offsets of the features from the datum.

Ordinate dimensions consist of an $X$ or $Y$ value with a leader line. $X$-datum ordinate dimensions measure the distance of a feature from the datum along the $X$ axis. $Y$-datum ordinate dimensions measure the distance along the $Y$ axis.
Locate the Datum

The location and orientation of the current UCS determines the ordinate values. Before creating ordinate dimensions, you typically set the UCS origin to coincide with the datum.

Locate the Leader

After you specify the feature location, you are prompted for the leader endpoint. By default, the leader endpoint that you specify automatically determines whether an X- or a Y-datum ordinate dimension is created. For example, you can create an X-datum ordinate dimension by specifying a location for the leader endpoint that is closer to vertical than horizontal.

After creating an ordinate dimension, you can easily relocate the dimension leader and text using grip editing. The dimension text is always aligned with the ordinate leader line.
Quick Reference

Commands

DIMORDINATE
  Creates ordinate dimensions.

QDIM
  Creates a series of dimensions quickly from selected objects.

UCS
  Manages user coordinate systems.

Create Arc Length Dimensions

Arc length dimensions measure the distance along an arc or polyline arc segment.

Typical uses of arc length dimensions include measuring the travel distance around a cam or indicating the length of a cable. To differentiate them from linear or angular dimensions, arc length dimensions display an arc symbol by default.

The arc symbol, also called a hat or cap, is displayed either above the dimension text or preceding the dimension text. The placement style can be changed on the New/Modify Dimension Style dialog box, Symbols and Arrows tab.

The extension lines of an arc length dimension can be orthogonal or radial.
NOTE  Orthogonal extension lines are displayed only when the included angle of the arc is less than 90 degrees.

Quick Reference

Commands
DIMARC
  Creates an arc length dimension.
DIMSTYLE
  Creates and modifies dimension styles.
PROPERTIES
  Controls properties of existing objects.

System Variables
DIMARCSYM
  Controls display of the arc symbol in an arc length dimension.

Modify Existing Dimensions

You can modify all components of the existing dimension objects in a drawing either individually or by using dimension styles.

Modify A Dimension

Dimensions can be modified to include more information than just the values of the dimension. Dimensions can also be modified visually by using breaks and by adjusting the spacing between them.

Overview of Modifying Dimensions

After you place a dimension, there are times when you need to modify the information that the dimension represents. You can add a jog line to a linear dimension to indicate that the dimension value does not represent the actual dimensioned value or add an inspection dimension to represent how often a dimension value of a manufactured part should be checked.
At times you might want to modify a dimension to simply improve readability. You can make sure that the extension or dimension lines do not obscure any objects; you can adjust the placement of linear dimensions so they are evenly spaced.

**Dimension Jog**

Jog lines are used to represent a dimension value that does not display the actual measurement in a linear dimension. Typically, the actual measurement value of the dimension is smaller than the displayed value.

The jog is made up of two parallel lines and a cross line that forms two 40-degree angles. The height of the jog is determined by the linear jog size value of the dimension style.

![Dimension Jog Diagram](image)

Once you add a jog to a linear dimension, you can position it by using grips. To reposition the jog, select the dimension and then select the grip. Move the grip to another point along the dimension line. You can also adjust the height of the jog symbol on a linear dimension on the Properties Inspector under Lines & Arrows.

**Quick Reference**

**Commands**

**DIMALIGNED**

Creates an aligned linear dimension.

**DIMBASELINE**

Creates a linear, angular, or ordinate dimension from the baseline of the previous or selected dimension.
DIMCONTINUE
Creates a dimension that starts from an extension line of a previously created dimension.

DIMJOGLINE
Adds or removes a jog line on a linear or aligned dimension.

DIMLINEAR
Creates a linear dimension.

DIMSTYLE
Creates and modifies dimension styles.

QDIM
Creates a series of dimensions quickly from selected objects.

**Inspection Dimension**

Inspection dimensions allow you to effectively communicate how frequently manufactured parts should be checked to ensure that the dimension value and tolerances of the parts are within the specified range.

When working with parts that need to meet a specific tolerance or dimension value before installing them into the final assembled product, you can use an inspection dimension to specify how often the part should be tested.

You can add an inspection dimension to any type of dimension object; it is composed of a frame and text values. The frame for an inspection dimension is made up of two parallel lines and the end is round or square. The text values are separated by vertical lines. An inspection dimension can contain up to three different fields of information: inspection label, dimension value, and inspection rate.
**Inspection Dimension Fields**

**Inspection Label** Text used to identify individual inspection dimensions. The label is located in the leftmost section of the inspection dimension.

**Dimension Value** Dimension value that is displayed is the same value before the inspection dimension is added. The dimension value can contain tolerances, text (both prefix and suffix), and the measured value. The dimension value is located in the center section of the inspection dimension.

**Inspection Rate** Text used to communicate the frequency that the dimension value should be inspected, expressed as a percentage. The rate is located in the rightmost section of the inspection dimension.

You can add inspection dimensions to any type of dimension. The current values of an inspection dimension are displayed on the Properties Inspector, under Misc. The values include the properties that are used to control the look of the frame, and the text for both the label and rate values.

**Quick Reference**

**Commands**

DIMINSPECT

Adds or removes inspection information for a selected dimension.

**Dimension Breaks**

With dimension breaks, you can keep the dimension, extension, or leader lines from appearing as if they are a part of the design.

Dimension breaks can be added to a dimension or a multileader automatically or manually. The method that you choose to place dimension breaks depends on the number of objects that intersect a dimension or multileader.
You can add dimension breaks to the following dimension and leader objects:

- Linear dimensions (aligned and rotated)
- Angular dimensions (2- and 3-point)
- Radial dimensions (radius, diameter, and jogged)
- Arc length dimensions
- Ordinate dimensions
- Multileaders (straight only)

The following dimension and leader objects do not support dimension breaks:

- Multileaders (spline only)
- “Legacy” leaders (straight or spline)

The following table explains the conditions where dimension breaks do not work or are not supported.

<table>
<thead>
<tr>
<th>Condition</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>No break in xrefs or blocks</td>
<td>Dimension breaks on dimensions or multileaders in xrefs and blocks are not supported. However, the objects in an xref or block can be used as the cutting edges for dimension breaks on dimensions or multileaders that are not in an xref or block.</td>
</tr>
<tr>
<td>No break on arrowhead and dimension text</td>
<td>Dimension breaks cannot be placed on an arrowhead or the dimension text. If you want a break to appear at the dimension text, it is recommended to use the background mask option. If the intersecting point of an object and the dimension are at the arrowhead or dimension text, the break will not be displayed until the intersecting object, or dimension or multileader are moved.</td>
</tr>
<tr>
<td>No break on trans-spatial dimensions</td>
<td>Automatic breaks are not supported for objects and dimensions or multileaders that are in different spaces. In order to break a dimension or multileader that is in a different space, you need to use the Manual option of the DIMBREAK command.</td>
</tr>
</tbody>
</table>
You can move dimension breaks from dimensions or multileaders. When removing dimension breaks from a dimension or multileader, all dimension breaks are removed. If there are some dimension breaks that you don’t want to remove, you need to add them again.

The following objects can be used as cutting edges when adding a dimension break:

- Dimension
- Leader
- Line
- Circle
- Arc
- Spline
- Ellipse
- Polyline
- Text
- Multiline text
- Blocks but limited to the previously mentioned objects in this list
- Xrefs but limited to the previously mentioned objects in this list

**Automatic Dimension Breaks**

To create an automatically placed dimension break, you select a dimension or multileader, and then use the Auto option of the DIMBREAK command. Automatic dimension breaks are updated any time the dimension or multileader, or intersecting objects are modified.

You control the size of automatically placed dimension breaks on the New/Modify Dimension Style dialog box, Symbols and Arrows tab. The specified size is affected by the dimension break size, dimension scale, and current annotation scale for the current viewport. For more information about annotation scaling, see *Scale Annotations* on page 656.

**Dimension Break Created by Selecting an Object**

Instead of placing a dimension break for each object that intersects a dimension or multileader, you can specify which of the intersecting objects to use.
Dimension breaks that are added by selecting individual intersecting objects are updated any time the dimension or multileader, or intersecting objects are modified.

**Dimension Break Created by Picking Two Points**

You can place a dimension break by picking two points on the dimension, extension, or leader line to determine the size and placement of the break. Dimension breaks that are added manually by picking two points are not automatically updated if the dimension or multileader, or intersecting object is modified.

So if a dimension or multileader with a manually added dimension break is moved or the intersecting object is modified, you might have to restore the dimension or multileader, and then add the dimension break again. The size of a dimension break that is created by picking two points is not affected by the current dimension scale or annotation scale value for the current viewport.

**Quick Reference**

**Commands**

DIMBREAK

Breaks or restores dimension and extension lines where they cross other objects.

DIMSTYLE

Creates and modifies dimension styles.

**Adjust Dimension Spacing**

You can automatically adjust existing parallel linear and angular dimensions in a drawing so they are equally spaced or aligned at the dimension line with each other.

Parallel linear and angular dimensions can be created in a number of different ways in a drawing. With the DIMLINEAR and DIMANGULAR commands you can place one dimension at a time; you can use the DIMBASELINE and DIMCONTINUE commands to help place additional linear dimensions based on the previous linear dimension placed.

The DIMBASELINE command uses the DIMDLI system variable to create equally spaced dimensions, but once the dimensions are placed, changing the value of the system variable has no affect on the spacing of dimensions. If
you change the text size or adjust the scale for the dimensions, they remain in the original position which can cause problems with overlapping dimension lines and text.

You can space linear and angular dimensions that overlap or are not equally spaced with the DIMSPACE command. The dimensions that are selected must be linear or angular, of the same type (rotated or aligned), parallel or concentric to one another, and on the extension lines of each other. You can also align linear and angular dimensions by using a spacing value of 0.

The following illustration shows parallel linear dimensions that are not equally spaced and then those that are equally spaced after using the DIMSPACE command.

Quick Reference

Commands

DIMALIGNED
Creates an aligned linear dimension.

DIMANGULAR
Creates an angular dimension.

DIMBASELINE
Creates a linear, angular, or ordinate dimension from the baseline of the previous or selected dimension.

DIMCONTINUE
Creates a dimension that starts from an extension line of a previously created dimension.

DIMLINEAR
Creates a linear dimension.
DIMSPACE

Adjusts the spacing between linear dimensions or angular dimensions.

DIMSTYLE

Creates and modifies dimension styles.

QDIM

Creates a series of dimensions quickly from selected objects.

System Variables

DIMDLI

Controls the spacing of the dimension lines in baseline dimensions.

Apply a New Dimension Style to Existing Dimensions

You can modify existing dimensions by applying a different dimension style. If you make changes to a dimension style, you can choose whether to update the dimensions associated with that dimension style.

When you create a dimension, the current dimension style is associated with that dimension. The dimension retains this dimension style unless you apply a new dimension style to it or set up dimension style overrides.

You can modify existing dimensions by applying a different dimension style. If you make changes to a dimension style, you can choose whether to update the dimensions associated with that dimension style.

You can restore an existing dimension style or apply the current dimension style, including any dimension style overrides, to selected dimensions.

Quick Reference

Commands

DIMOVERRIDE

Controls overrides of system variables used in selected dimensions.

DIMSTYLE

Creates and modifies dimension styles.
PROPERTIES
Controls properties of existing objects.

System Variables
DIMCLRD
Assigns colors to dimension lines, arrowheads, and dimension leader lines.

Override a Dimension Style
With dimension style overrides, you can temporarily change a dimensioning
system variable without changing the current dimension style.

A dimension style override is a change made to specific settings in the current
dimension style. It is equivalent to changing a dimensioning system variable
without changing the current dimension style.

You can define dimension style overrides for individual dimensions, or for
the current dimension style.

■ For individual dimensions, you may want to create overrides to suppress
a dimension’s extension lines or modify text and arrowhead placement so
that they do not overlap drawing geometry without creating a different
dimension style.

■ You can also set up overrides to the current dimension style. All dimensions
you create in the style include the overrides until you delete the overrides,
save the overrides to a new style, or set another style current. For example,
if you choose Override in the Dimension Style Manager, and change the
color of extension lines on the Override Current Style dialog box, Lines
tab, the current dimension style remains unchanged. However, the new
value for color is stored in the DIMCLRE system variable. The next
dimension you create will have extension lines in the new color. You can
save the dimension style overrides as a new dimension style.

Some dimension characteristics are common to a drawing or to a style of
dimensioning and are therefore suited to be permanent dimension style
settings. Others generally apply on an individual basis and can be applied
more effectively as overrides. For example, a drawing usually uses a single type
of arrowhead, so it makes sense to define the arrowhead type as part of the
dimension style. Suppression of extension lines, however, usually applies in
individual cases only and is more suited to a dimension style override.

828 | Chapter 29  Dimensions and Tolerances
There are several ways to set up dimension style overrides. You can change options in the dialog boxes or change system variable settings at the Command prompt. You reverse the override by returning the changed settings to their original values. The overrides apply to the dimension you are creating and all subsequent dimensions created with that dimension style until you reverse the override or make another dimension style current.

**Example: Change a Dimension Style Override at the Command Prompt**

You can override the current dimension style while creating a dimension by entering the name of any dimensioning system variable at any prompt. In this example, the dimension line color is changed. The change affects subsequent dimensions you create until you reverse the override or make another dimension style current.

Command: `dimoverride`

Enter dimension variable name to override or [Clear overrides]: `dimclrd`

Enter new value for dimension variable <BYBLOCK>: `5`

Enter dimension variable name to override: `Enter another dimension variable name or press Enter`

Select objects: *Use an object selection method and press Enter when you finish*

**Quick Reference**

**Commands**

- **DIMOVERRIDE**
  Controls overrides of system variables used in selected dimensions.

- **DIMSTYLE**
  Creates and modifies dimension styles.

- **PROPERTIES**
  Controls properties of existing objects.

**System Variables**

- **DIMCLRD**
  Assigns colors to dimension lines, arrowheads, and dimension leader lines.
Modify Dimension Text

Once you've created a dimension, you can change the location and orientation of the existing dimension text or replace it with new text.

Once you've created a dimension, you can rotate the existing text or replace it with new text. You can move the text to a new location or back to its home position, which is the position defined by the current dimension style. In the following illustration, the home position is above and centered on the dimension line.

When you rotate or replace dimension text, you specify the change first, for example, rotating the text to be at an angle. When you move dimension text, you select a single dimension to move.

You can move dimension text to the left, right, or center along the dimension line or to any position inside or outside the extension lines. A quick and simple way to do this is by using grips. If you move text up or down, the current vertical alignment of the text relative to the dimension line is not changed, so the dimension and extension lines are modified accordingly. The following illustration shows the result of moving text down and to the right. The text remains centered vertically in relation to the dimension line.
See also:

- Control Dimension Text on page 783

**Quick Reference**

**Commands**

**DDEDIT**
Edits single-line text, dimension text, attribute definitions, and feature control frames.

**DIMEDIT**
Edits dimension text and extension lines.

**DIMTEDIT**
Moves and rotates dimension text and relocates the dimension line.

**PROPERTIES**
Controls properties of existing objects.

**System Variables**

**DIMCLRT**
Assigns colors to dimension text.

**DIMDSEP**
Specifies a single-character decimal separator to use when creating dimensions whose unit format is decimal.

**DIMJUST**
Controls the horizontal positioning of dimension text.

**DIMTAD**
Controls the vertical position of text in relation to the dimension line.

**DIMTXTDIRECTION**
Specifies the reading direction of the dimension text.

**DIMTIH**
Controls the position of dimension text inside the extension lines for all dimension types except Ordinate.
DIMTMOVE
Sets dimension text movement rules.

DIMTOH
Controls the position of dimension text outside the extension lines.

DIMTVP
Controls the vertical position of dimension text above or below the dimension line.

DIMUPT
Controls options for user-positioned text.

**Modify Dimension Geometry**

Grip editing is the quickest and easiest way to modify the location of dimension elements. How you edit dimensions depends whether the dimension is associative.

You can modify dimensions with the editing commands and with grip editing. Grip editing is the quickest and easiest way to modify dimensions. How you edit dimensions depends on whether the dimension is associative.

**Modify Associative Dimensions**

Associative dimensions retain their associativity to dimensioned objects through many editing commands if both the dimension and the associated geometry are selected and operated on with a single command. For example, if a dimension and its associated geometry are moved, copied, or arrayed in the same command, each dimension retains associativity with its respective geometry.

In some circumstances, dimensions are automatically disassociated, including

- If the associated geometric object is erased
- If the associated geometric object undergoes a boolean operation such as UNION or SUBTRACT
- If grip editing is used to stretch a dimension parallel to its dimension line
- If the association to a geometric object is specified using the Apparent Intersection object snap, and the geometric object is moved so that the apparent intersection no longer exists
In other circumstances, a dimension may become partially associated. For example, if a linear dimension is associated with the endpoints of two geometric objects and one of the objects is erased, the remaining association is preserved. The disassociated end of the linear dimension may then be associated with another geometric object using DIMREASSOCIATE.

**NOTE** The Command prompt displays a warning message if a dimension is disassociated.

**Modify Non-associative Dimensions**

For non-associative dimensions, when you edit dimensioned objects, you must include the relevant dimension definition points in the selection set, or the dimension is not updated. Definition points determine the dimension location. For example, to stretch a dimension, you must include the appropriate definition points in the selection set. You can easily include them by turning on grips and selecting the object so that the grips are highlighted.

The definition points for each type of dimension are indicated in the following illustrations. The middle point of the dimension text is a definition point for all dimension types.
If no angle vertex is shown, definition points are placed at the ends of the lines that form the angle. In the two-line angular example, a definition point is placed at the center point of the dimensioned arc.

**NOTE** Definition points are drawn on a special layer named DEFPOINTS, which is not printed.

**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 775

Quick Reference

Commands

**DIMEDIT**
Edits dimension text and extension lines.

**DIMDISASSOCIATE**
Removes associativity from selected dimensions.

**DIMREASSOCIATE**
Associates or reassociates selected dimensions to objects or points on objects.

**EXPLODE**
Breaks a compound object into its component objects.

**STRETCH**
Stretches objects crossed by a selection window or polygon.

System Variables

**DIMASSOC**
Controls the associativity of dimension objects and whether dimensions are exploded.

Change Dimension Associativity

You may need to change the associativity of dimensions in several circumstances including adding associativity to dimensions created in previous releases.

You may need to change the associativity of dimensions in several circumstances such as the following:

- Redefine the associativity of dimensions in drawings that have been edited significantly.
- Add associativity to dimensions that have been partially disassociated.
Add associativity to dimensions in legacy drawings.

Remove associativity from dimensions in drawings that will be used by people working in releases prior to AutoCAD 2002, but who do not want any proxy objects in the drawings.

Reassociate Dimensions to Different Objects

With DIMREASSOCIATE, you can select one or more dimensions and step through the extension-line origin points of each dimension. For each extension-line origin point, you can specify a new association point on a geometric object. Association points determine the attachment of extension lines to locations on geometric objects.

**NOTE** When you create or modify associative dimensions, it is important to locate their association points carefully so that if you make a future design change, the geometric objects that you change will also change the dimensions associated with them.

When you use the DIMREASSOCIATE command, a marker is displayed that indicates whether each successive extension line origin point of the dimension is associative or nonassociative. A square with an X in it means that the point is associated with a location on an object, while an X without the square means that the point is not associated with an object. Use an object snap to specify the new association for the extension-line origin point or press Enter to skip to the next extension-line origin point.

**NOTE** The marker disappears if you pan or zoom.

Change Non-associative Dimensions to Associative

You can change all the non-associative dimensions in a drawing to associative. Select all non-associative dimensions, and then use DIMREASSOCIATE to step through the dimensions, associating each one with locations on geometric objects.

Change Associative Dimensions to Non-associative

You can change all associative dimensions in a drawing to nonassociative dimensions. Select all associative dimensions, and then use DIMDISASSOCIATE to convert them into nonassociative dimensions.

See also:

- Associative Dimensions on page 771
Quick Reference

Commands

DIMDISASSOCIATE
Removes associativity from selected dimensions.

DIMREASSOCIATE
Associates or reassociates selected dimensions to objects or points on objects.

DIMREGEN
Updates the locations of all associative dimensions.

EXPLODE
Breaks a compound object into its component objects.

System Variables

DIMASSOC
Controls the associativity of dimension objects and whether dimensions are exploded.

Add Geometric Tolerances
You can add geometric tolerances that show acceptable deviations of form, profile, orientation, location, and runout of a feature.

Overview of Geometric Tolerances
Geometric tolerances show acceptable deviations of form, profile, orientation, location, and runout of a feature.

You add geometric tolerances in feature control frames. These frames contain all the tolerance information for a single dimension. Geometric tolerances can be created with or without leader lines, depending on whether you create them with TOLERANCE or LEADER.

A feature control frame consists of two or more components. The first feature control frame contains a symbol that represents the geometric characteristic.
to which a tolerance is being applied, for example, location, profile, form, orientation, or runout. Form tolerances control straightness, flatness, circularity and cylindricity; profiles control line and surface. In the illustration, the characteristic is position.

You can use most editing commands to change feature control frames, and you can snap to them using the object snap modes. You can also edit them with grips.

**NOTE** Unlike dimensions and leaders, geometric tolerances cannot be associated with geometric objects.

You can also create tolerances. For more information about creating and working with an annotative tolerances, see Create Annotative Dimensions and Tolerances on page 664.

**See also:**
- Scale Annotations on page 656
- Create Annotative Dimensions and Tolerances on page 664
Quick Reference

Commands

LEADER

Creates a line that connects annotation to a feature.

TOLERANCE

Creates geometric tolerances contained in a feature control frame.

Material Conditions

Material conditions apply to features that can vary in size.

The second compartment contains the tolerance value. Depending on the control type, the tolerance value is preceded by a diameter symbol and followed by a material condition symbol.

Material conditions apply to features that can vary in size:

- At maximum material condition (symbol M, also known as MMC), a feature contains the maximum amount of material stated in the limits.

- At MMC, a hole has minimum diameter, whereas a shaft has maximum diameter.

- At least material condition (symbol L, also known as LMC), a feature contains the minimum amount of material stated in the limits.

- At LMC, a hole has maximum diameter, whereas a shaft has minimum diameter.

- Regardless of feature size (symbol S, also known as RFS) means a feature can be any size within the stated limits.
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.

![Datum Reference Frame Illustration]

Quick Reference

Commands

LEADER
  Creates a line that connects annotation to a feature.

TOLERANCE
  Creates geometric tolerances contained in a feature control frame.

Projected Tolerance Zones

Projected tolerances are used to make the tolerance more specific.

Projected tolerances are specified in addition to positional tolerances to make the tolerance more specific. For example, projected tolerances control the perpendicularity tolerance zone of an embedded part.

The symbol for projected tolerance ( ) is preceded by a height value, which specifies the minimum projected tolerance zone. The projected tolerance zone
height and symbol appear in a frame below the feature control frame, as shown in the following illustration.

Quick Reference

Commands

LEADER

Creates a line that connects annotation to a feature.

TOLERANCE

Creates geometric tolerances contained in a feature control frame.

Composite Tolerances

A composite tolerance specifies two tolerances for the same geometric characteristic of a feature or for features that have different datum requirements. One tolerance relates to a pattern of features and the other tolerance to each feature within the pattern. The individual feature tolerance is more restrictive than the pattern tolerance.

In the following illustration, the point where datums A and B intersect is called the datum axis, the point from which the position of the pattern is calculated.

A composite tolerance could specify both the diameter of the pattern of holes and the diameter of each individual hole, as in the following illustration.
When you add composite tolerances to a drawing, you specify the first line of a feature control frame and then choose the same geometric characteristic symbol for the second line of the feature control frame. The geometric symbol compartment is extended over both lines. You can then create a second line of tolerance symbols.

**Quick Reference**

**Commands**

**LEADER**

Creates a line that connects annotation to a feature.

**TOLERANCE**

Creates geometric tolerances contained in a feature control frame.
Plot and Publish Drawings
Specify Settings for Plotting

Before you plot a drawing, you must specify the settings that determine the output. To save time, you can store these settings with the drawing as a named page setup.

Save Plot Settings as Named Page Setups

If you want to plot the same layout more than one way, or if you want to specify the same output options for several layouts, use named page setups.

Before you plot a drawing, you must specify the settings that determine the appearance and format of the output. To save time, you can store these settings with the drawing as a named page setup.

For example, when you access a layout for the first time, a single layout viewport is displayed, and a dashed line indicates the printable area of the paper for the currently configured paper size and printer or plotter.
In addition, the page setup also includes many other settings and options such as

- The orientation of the plot, portrait or landscape
- The plot scale
- Whether lineweights should be plotted
- The shading style

By default, the first time you access a layout, it becomes initialized, and a default page setup is assigned to it. Default page setups are assigned names such as *model*, *layout1*, *layout2*, and so on.

Quick Reference

Commands

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

PLOT
   Outputs a drawing to a printer or file.

PSETUPIN
   Imports a user-defined page setup into a new drawing layout.

Reuse Named Page Setups

You can save plot device and other page setup settings as named page setups that can be modified and imported into other drawings.

Named page setups are saved in the current drawing file and can be imported into other drawing files and applied to other layouts.

If you want to plot the same layout more than one way, or if you want to specify the same output options for several layouts, use named page setups.

You can apply a named page setup to model space or to a layout using the Page Setup Manager. Other options available in the Page Setup Manager include

- Apply a named page setup saved with one layout to another layout in the same drawing
Modify the settings of a page setup at any time

Import a named page setup from another drawing, and apply it to layouts in the current drawing

You can also apply different named page setups to the same layout to achieve specific results when plotting. For example, you might create the named page setups in the following table to control scaling and paper size.

<table>
<thead>
<tr>
<th>Page setup name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NoScaling</td>
<td>Plot at scale 1:1, E-size sheet</td>
</tr>
<tr>
<td>Scale 1 to 2</td>
<td>Plot at scale 1:2, C-size sheet</td>
</tr>
<tr>
<td>Draft</td>
<td>Plot to the draft-quality plotter</td>
</tr>
<tr>
<td>Final</td>
<td>Plot to the high-quality plotter</td>
</tr>
<tr>
<td>Fit-to-Paper</td>
<td>Fit to Paper, A-size sheet</td>
</tr>
</tbody>
</table>

Once you specify a named page setup for a layout, whenever you plot the layout, it is plotted with the settings you specified.

Quick Reference

Commands

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

PLOT
Outputs a drawing to a printer or file.

PSETUPIN
Imports a user-defined page setup into a new drawing layout.
Specify Page Setup Settings

Page setups are associated with model space and with layouts, and are saved in the drawing file. The settings specified in a page setup determine the appearance and format of your final output.

Select a Printer or Plotter for a Layout

To print a layout, select a printing or plotting device in the Page Setup dialog box. You can also view details about the name and location of the device, and change the device’s configuration.

The printer or plotter you select in the Page Setup dialog box determines the printable area of the layout. This printable area is indicated by the dashed line in the layout. If you change the paper size or the printing or plotting device, it may change the printable area of your drawing page.

See also:
- Select a Printer or Plotter on page 860

Quick Reference

Commands

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

PLOT
- Outputs a drawing to a printer or file.

PSETUPIN
- Imports a user-defined page setup into a new drawing layout.

Select a Paper Size for a Layout

You can select a paper size from a standard list, or you can add custom paper sizes using the Page Setup dialog box or Print dialog box.

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.

**Quick Reference**

**Commands**

PAGESETUP

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

PLOT

Outputs a drawing to a printer or file.

---

**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.
Quick Reference

Commands

PAGESETUP

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

PLOT

Outputs a drawing to a printer or file.

Set the Plot Area of a Layout

You can specify the plot area to determine what will be included in the plot.

When you prepare to plot from model space or a layout, you can specify the plot area to determine what will be included in the plot. When you create a new layout, the default Plot Area option is Layout. Layout plots all objects within the printable area of the specified paper size.

The Display Plot Area option plots all the objects displayed in the drawing. The Extents Plot Area option plots all the visible objects in the drawing. The View Plot Area option plots a saved view. You can use the Window Plot Area option to define an area to be plotted.

See also:

- Specify the Area to Plot on page 861

Quick Reference

Commands

PAGESETUP

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

PLOT

Outputs a drawing to a printer or file.
Adjust the Plot Offset of a Layout

The printable area of a drawing sheet is defined by the selected output device and is represented by the dashed line in a layout. When you change the output device, the printable area may change.

The plot offset specifies an offset of the plot area relative to the lower-left corner (the origin) of the printable area or the edge of the paper.

You can offset the geometry on the paper by entering a positive or negative value in the X and Y Offset boxes. However, this may result in the plot area being clipped.

If you choose to plot an area other than the entire layout, you can also center the plot on the sheet of paper.

Quick Reference

Commands

PAGESETUP

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

PLOT

Outputs a drawing to a printer or file.
System Variables

PLOTOFFSET

Controls whether the plot offset is relative to the printable area or to the edge of the paper.

Set the Plot Scale for a Layout

When you plot a drawing layout, you can either specify a precise scale for the layout or fit the image to the paper.

Normally, you plot a layout at a 1:1 scale. To specify a different scale for the layout, set the plot scale for the layout in the Page Setup or the Plot dialog box. In those dialog boxes, you can select a scale from a list or enter a scale.

NOTE You can modify the list of scales with SCALELISTEDIT.

When you are reviewing an early draft view, a precise scale is not always important. You can use the Fit to Paper setting to plot the layout at the largest possible size that fits the paper.

See also:

- Scale Views in Layout Viewports on page 147
- Draw, Scale, and Annotate in Model Space on page 131

Quick Reference

Commands

PAGESETUP

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

PLOT

Outputs a drawing to a printer or file.

SCALELISTEDIT

Controls the list of scales available for layout viewports, page layouts, and plotting.
Set the Lineweight Scale for a Layout

You can scale lineweights proportionately in a layout with the plot scale.

Typically, lineweights specify the line width of plotted objects and are plotted with the line width size regardless of the plot scale. Most often, you use the default plot scale of 1:1 when plotting a layout. However, if you want to plot an E-size layout that is scaled to fit on an A-size sheet of paper, for example, you can specify lineweights to be scaled in proportion to the new plot scale.

See also:

■ Control Lineweights on page 186

Quick Reference

Commands

PAGESETUP
Controls the page layout, plotting device, paper size, and other settings for each new layout.
PLOT
Outputs a drawing to a printer or file.

Select a Plot Style Table for a Layout

A plot style table is a collection of plot styles assigned to a layout or model space.

A plot style is an object property, similar to linetype and color. Therefore, it can be assigned to an object or a layer and they control an object’s plotted properties.

If you select the Display Plot Styles option under Plot Style Table (Pen Assignments), the properties of the plot styles assigned to objects are displayed in the selected layout.

See also:

■ Control How Objects Are Plotted on page 865
Quick Reference

Commands

PAGESETUP

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

PLOT

Outputs a drawing to a printer or file.

Set Shaded Viewport and Plot Options for a Layout

Shaded viewport and plot options settings affect how objects are plotted and are saved in the page setup.

Shaded viewport and plot options affect how objects are plotted. The options for shaded viewport plotting give you a large degree of flexibility in conveying your three-dimensional designs to others. You can convey your design intent by choosing how viewports are plotted and by specifying resolution levels.

Shaded Viewport Plotting Options

With shaded plotting options, you can choose whether to plot a set of shaded objects using the As Displayed, Wireframe, Hidden, or Rendered option.

With shaded plotting options, you can choose whether to plot a set of shaded objects using the As Displayed, Wireframe, or Hidden option.

Shaded viewport plotting options apply to all objects in viewports and model space. If you use the Shaded 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.

Shaded viewport plotting options apply to all objects in viewports and model space. If you use the Shaded option, plot style tables included in the page setup do not affect plots.

NOTE  Shaded viewport plotting requires a raster-capable device. Most modern plotters and printers are raster-capable devices.

See also:

■  Set Shaded Viewport Options on page 867
Quick Reference

Commands

PAGESETUP

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

PLOT

Outputs a drawing to a printer or file.
Print or Plot Drawings

Once you have completed a drawing, you can use a number of methods to output the drawing. You can plot the drawing on paper or create a file for use with another application. In either case, you select the plot settings.

Overview of Plotting

Understanding terms and concepts that relate to plotting makes your first plotting experience in the program easier.

Am I Printing or Plotting?

The terms printing and plotting can be used interchangeably for CAD output. Historically, printers would generate text only, and plotters would generate vector graphics. As printers became more powerful and could generate high-quality raster images of vector data, the distinction mostly disappeared.

In addition to paper output, electronic delivery of multiple drawing sheets uses the encompassing term, publishing. Also, the capability to generate physical models in plastic and metal is called 3D printing.

Layouts

A layout represents a drawing sheet, and typically includes

- A drawing border and title block
- One or more layout viewports that display views of model space
- General notes, labels, and possibly dimensions
- Tables and schedules
Usually a drawing file contains only one layout, but you can create as many layouts as you need. The first time you display a layout, it is initialized and a default page setup is assigned to it.

**Page Setups**

When you create a layout, you specify a plotter, and settings such as paper size and orientation. These settings are saved in the drawing as a page setup. Each layout can be associated with a different page setup.

You can control these settings for layouts and for model space using the Page Setup Manager. You can name and save page setups for use with other layouts.

If you do not specify all the settings in the Page Setup dialog box when you create a layout, you can set up the page just before you plot. Or you can override a page setup at plot time. You can use the new page setup temporarily for the current plot, or you can save the new page setup.

**Plot Styles**

A plot style is an optional method that controls how each object or layer is plotted. Assigning a plot style to an object or a layer overrides properties such as color, lineweight, and linetype when plotting. Only the appearance of plotted objects is affected by plot style.

Plot style tables collect groups of plot styles, and save them in a file that you can later apply when plotting.

The Plot Style Manager is a folder that contains all the available plot style tables, along with the Add-A-Plot-Style wizard.

**NOTE** The Plot Style Manager is not available on the Mac. You can plot drawings with plot styles defined in them, but you cannot modify the plot styles or create new ones.

There are two plot style types: color-dependent and named. A drawing can use only one type of plot style table. You can convert a plot style table from one type to the other. You can also change the type of plot style table a drawing uses once it has been set.

For color-dependent plot style tables, an object’s color determines how it is plotted. These plot style table files have .ctb extensions. You cannot assign color-dependent plot styles directly to objects. Instead, to control how an object is plotted, you change its color. For example, all objects assigned the color red in a drawing are plotted the same way.
Named plot style tables use plot styles that are assigned directly to objects and layers. These plot style table files have .stb extensions. Using them enables each object in a drawing to be plotted differently, independent of its color.

Plot Stamps

A plot stamp is a line of text that is added to your plot. You can specify where this text is located on the plot in the Plot Stamp dialog box. Turn this option on to add specified plot stamp information—including drawing name, layout name, date and time, and so on—to a drawing that is plotted to any device. You can choose to record the plot stamp information to a log file instead of plotting it, or in addition to plotting it.

IMPORTANT A drawing file or drawing template file that was created with an educational version will always be plotted with the following plot stamp: PRODUCED BY AN AUTODESK EDUCATIONAL PRODUCT. Blocks and xrefs created with an educational version and used in a commercial version will also result in the educational plot stamp being plotted.

See also:
- Create Multiple-View Drawing Layouts (Paper Space) on page 137
- Specify Settings for Plotting on page 845
- “Use Plotters and Printers” in the Driver and Peripheral Guide

Quick Reference

Commands

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

PLOT
Outputs a drawing to a printer or file.

PLOTSTAMP
Places a plot stamp on a specified corner of each drawing and logs it to a file.

VIEWPLOTDETAILS
Displays information about completed print jobs.
System Variables
BACKGROUNDPLOT

Controls whether background plotting is turned on or off for plotting and publishing.

Use a Page Setup to Specify Plot Settings

You can use a page setup to save and reuse settings for your plot jobs.

When you select a page setup in the Plot dialog box, the settings from the page setup are added to the Plot dialog box. You can choose to plot with those settings, or change any of the settings individually and then plot.

Any settings you specify in the Plot dialog box can be saved as a new named page setup by clicking the Add button in the Page Setup area.

Any settings specified in the Plot dialog box, whether you've applied a page setup from the Page Setup list, or changed the settings individually, can be saved to the layout for use the next time you plot.

See also:
- Specify Settings for Plotting on page 845
- “Use Plotters and Printers” in the Driver and Peripheral Guide

Quick Reference

Commands

PAGESETUP

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

PLOT

Outputs a drawing to a printer or file.

Select a Printer or Plotter

Before plotting a drawing, you must select a printer or plotter. The device you select affects the printable area of the drawing.
After selecting a printing or plotting device, you also can easily plot a drawing using the default settings in the Plot dialog box.

See also:

■ Specify Settings for Plotting on page 845
■ “Use Plotters and Printers” in the Driver and Peripheral Guide

Quick Reference

Commands
PAGESETUP
Controls the page layout, plotting device, paper size, and other settings for each new layout.
PLOT
Outputs a drawing to a printer or file.

Specify the Area to Plot

When plotting a drawing, you must specify the area of the drawing to plot. The Print dialog box provides the following options under What to Print.

■ 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.

■ 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.
Quick Reference

Commands

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

PLOT
Outputs a drawing to a printer or file.

Set Paper Size

When plotting a drawing, select the paper size that you want to use.

If you plot from a layout, you may have already specified a paper size in the Page Setup dialog box. However, if you plot from the model space, you need to specify a paper size when you plot.

In the Print 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 printers or plotters includes all those that are currently configured through the Mac operating system.

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.
**System Variables**

**PAPERUPDATE**

Controls the display of a warning dialog box when attempting to print a layout with a paper size different from the paper size specified by the default for the plotter configuration file.

**Position the Drawing on the Paper**

There are several ways to position a drawing on the paper. You can specify the printable area, set the position of the plot, and set the orientation.

**Specify the Printable Area**

The printable area is displayed by a dashed border in a layout. The plotter and paper size you select determine the printable area.

![Dashed border in a layout](image)

**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.

**Quick Reference**

**Commands**

**PAGESETUP**

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

Outputs a drawing to a printer or file.

Set the Position of the Plot

The printable area of a drawing sheet is defined by the selected printer or plotter, but you can change the position of plot relative to the printable area or to the edge of the paper.

You can specify an offset of the plot area relative to the lower-left corner (the origin) of the printable area.

NOTE If you are plotting from model space, the settings for this option are located in the Print dialog box, Advanced settings, Plot Offset area. If you are plotting from a layout, the settings are located in the Print dialog box, Edit Page Setup, Advanced settings.

You can shift the drawing on the paper by entering positive or negative values in the X and Y boxes. However, this can result in the plot area being clipped. If the Plot Area is not set to Layout (Extents, Display, or Window), you can also select the Center the Plot option.

NOTE If you specify a different printer or plotter, the printable area might change.

Quick Reference

Commands

PAGESETUP

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

PLOT

Outputs a drawing to a printer or file.

Set Drawing Orientation

The drawing orientation determines whether the position of the plotted drawing is landscape (the longer edge of the drawing is horizontal) or portrait (the longer edge of the drawing is vertical). This is based on the size of paper selected. You can also choose to plot upside down.
Quick Reference

Commands

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

PLOT
Outputs a drawing to a printer or file.

System Variables

PLOTROTMODE
Controls the orientation of plots.

Control How Objects Are Plotted

You can control how objects are plotted by setting the plot scale, by using plot styles and plot style tables, and by setting an object's layer properties.

Set Plot Scale

When you specify a scale to output your drawing, you can choose from a list of real-world scales, enter your own scale, or select Fit to Paper to scale the drawing to fit onto the selected paper size.

Usually, you draw objects at their actual size. That is, you decide how to interpret the size of a unit (an inch, a millimeter, a meter) and draw on a 1:1 scale. For example, if your unit of measurement is millimeters, then every unit in your drawing represents a millimeter. When you plot the drawing, you either specify a precise scale or fit the image to the paper.

Most final drawings are plotted at a precise scale. The method used to set the plot scale depends on whether you plot model space or a layout:

- From model space, you can establish the scale in the Plot dialog box. This scale represents a ratio of plotted units to the world-size units you used to draw the model.

- In a layout, you work with two scales. The first affects the overall layout of the drawing, which usually is scaled 1:1, based on the paper size. The second is the scale of the model itself, which is displayed in layout.
viewports. The scale in each of these viewports represents a ratio of the paper size to the size of the model in the viewport.

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

Set a Specific Scale

When you plot, the paper size you select determines the unit type, inches or millimeters. For example, if the paper size is in mm, entering 1 under mm and 10 under Units produces a plotted drawing in which each plotted millimeter represents 10 actual millimeters.

The illustrations show a light bulb plotted at three different scales.

Scale the Drawing to Fit the Page

When you review drafts, a precise scale is not always important. You can use the Fit to Paper option to plot the view at the largest possible size that fits the paper. The height or width of the drawing is fit to the corresponding height or width of the paper.

When you plot a perspective view from model space, the view is scaled to fit the paper even when you enter a scale.

When you select the Fit to Paper option, the text boxes change to reflect the ratio of plotted units to drawing units. This scale is updated whenever you change the paper size, plotter, plot origin, orientation, or size of the plotted area in the Plot dialog box.

NOTE This option is not available when the Plot Area is set to Layout.
Quick Reference

Commands

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

PLOT
  Outputs a drawing to a printer or file.

SCALELISTEDIT
  Controls the list of scales available for layout viewports, page layouts, and plotting.

Set Shaded Viewport Options

You can choose among several options for plotting shaded and rendered viewports. You can plot a viewport as it is displayed, in wireframe, with hidden lines removed, or as rendered.

Overview of Shaded Viewport Plotting

The options for shaded viewport plotting provide flexibility in presenting your 3D designs to others.

With shaded plotting options, you can choose whether to plot a set of shaded objects as displayed or in wireframe, hidden mode, a visual style, or rendered.

Shaded viewport plotting options apply to all objects in viewports and model space. If you use the Shaded or Rendered options, plot style tables included in the page setup do not affect plots. If you use the Render option, two-dimensional wireframe objects, such as lines, arcs, and text, are not plotted.

NOTE Shaded viewport plotting requires a raster-capable device. Most modern plotters and printers are raster-capable devices.
Quick Reference

Commands

3DCONFIG
Sets options that affect 3D display performance.

PLOT
Outputs a drawing to a printer or file.

SHADEMODE
Starts the VSCURRENT command.

VSCURRENT
Sets the visual style in the current viewport.

Specify Shaded Plotting Settings

If you are plotting a drawing that contains 3D solids that are shaded, you can control how the drawing is plotted.

Specifically, you can choose from the following options:

- **As Displayed.** Plots the design as it is displayed; all the shading is preserved.
- **Wireframe.** Displays lines and curves to represent object boundaries.
- **Hidden.** Suppresses the plotting of objects that are located behind other objects.
- **Visual Styles.** Plots the design as it appears in the visual style you select.
- **Rendered.** Renders objects before they are plotted, based on Render options you set before you plot or based on the render preset you select.
- **Render Presets.** Renders objects based on the render preset you select.

You can select an option for your drawing either from model space or from a layout. From model space, the options are available in the Properties Inspector palette and the Print dialog box. From a layout, after you select a viewport, the options are available from the shortcut menu and from the Properties Inspector palette.
NOTE If you select the Rendered option, specify Render settings before plotting. If the Rendered option is used for a highly complex set of objects, the hardcopy output might contain only the viewport border.

Quick Reference

Commands

3DCONFIG

Sets options that affect 3D display performance.

PLOT

Outputs a drawing to a printer or file.

SHADEMODE

Starts the VSCURRENT command.

VSCURRENT

Sets the visual style in the current viewport.

Specify a Resolution Level for Shaded Plotting

You can set the resolution of shaded plots for either greater speed or higher fidelity.

After you select an appropriate plotter, you can specify the level of quality for plotted output. The quality level determines the dots per inch (dpi). The dpi that corresponds to a quality level is based on the plotter you select.

The maximum dpi available is also based on the plotter you select. You can specify a custom quality level and directly change the dpi to a setting between 100 and the maximum dpi of the plotter.

The higher the fidelity, the more computer memory is used, so the longer it takes to plot. High fidelity is not necessary for all plots, and a setting between 300 and 600 dpi is generally sufficient for most plots.
Quick Reference

Commands
PLOT

Outputs a drawing to a printer or file.

Set Options for Plotted Objects

In the Print and the Page Setup dialog boxes, you can choose from options that affect how objects are plotted.

- **Shaded Viewport Plotting.** Specifies shaded plotting options: As Displayed, Wireframe, or Hidden. The effect of this setting is reflected in the plot preview, but not in the layout.

- **Plot Object Lineweights.** Specifies that lineweights assigned to objects and layers are plotted.

- **Plot Transparency.** Specifies that transparency levels applied to objects and layers are plotted. Plot Transparency applies to wireframe and hidden plots only. Other visual styles, such as Realistic, Conceptual, or Shaded will always plot with transparency.
  
  IMPORTANT This setting can be overridden by the PLOTTRANSPARENCYOVERRIDE system variable.

- **Plot with Plot Styles.** Specifies that the drawing is plotted using plot styles. Selecting this option automatically plots lineweights. If you do not select this option, objects are plotted with their assigned properties and not with the plot style overrides.

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

- **Plot Paper Space Last.** Specifies that objects in model space are plotted before those in paper space.

- **Hide Paperspace Objects.** Specifies whether the Hide operation applies to objects in the layout viewport. The effect of this setting is reflected in the plot preview, but not in the layout.

- **Plot Stamp On.** Turns on plot stamps and places a plot stamp on a specified corner of each drawing and can add it to a log file. Plot stamp settings are
specified in the Plot Stamp dialog box, where you can specify the information you want applied to the plot stamp, such as drawing name, date and time, plot scale, and so on.

■ **Save Changes to Layout.** Saves changes you make in the Print dialog box to the layout.

## Quick Reference

### Commands

**LWEIGHT**

Sets the current lineweight, lineweight display options, and lineweight units.

**PAGESETUP**

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

**PLOT**

Outputs a drawing to a printer or file.

**PLOTSTAMP**

Places a plot stamp on a specified corner of each drawing and logs it to a file.

**PROPERTIES**

Controls properties of existing objects.

## Use Plot Styles to Control Plotted Objects

You can control many aspects of how an object is plotted by using plot styles.

## Overview of Plot Styles

**NOTE** The Plot Style Manager is not available on the Mac. You can plot drawings with plot styles defined in them, but you cannot modify the plot styles or create new ones.
A plot style is an object property, similar to linetype and color. A plot style can be assigned to an object or assigned to a layer. A plot style controls an object’s plotted properties, including

- Color
- Dither
- Grayscale
- Pen number
- Virtual pen
- Screening
- Linetype
- Lineweight
- Transparency
- Line end style
- Line join style
- Fill style

Using plot styles gives you great flexibility because you can set them to override other object properties or turn off the override as needed.

Groups of plot styles are saved in either of two types of plot style tables: color-dependent (CTB) or named (STB). Color-dependent plot style tables set style based on the color of the object. Named plot styles can be assigned to an object independent of color.

**NOTE** Plot styles are not available for objects with the Jitter edge modifier applied (VISUALSTYLES).

**Quick Reference**

**Commands**

PAGESETUP

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

CPlotStyle

Controls the current plot style for new objects.

DefPlStyle

Specifies the default plot style for all layers in a drawing when opening a
drawing that was created in a release prior to AutoCAD 2000, or for Layer 0
when creating a new drawing from scratch without using a drawing template.

DefPlStyle

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

PStyleMode

Indicates whether the current drawing is in a Color-Dependent or Named
Plot Style mode.

PStylePolicy

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

Choose a Type of Plot Style Table

A plot style table is a collection of plot styles assigned to a layout or the Model
tab. There are two types of plot style tables: color-dependent plot style tables
and named plot style tables.

NOTE On the Mac, you cannot specify a plot style table for new drawings. You
can plot drawings with plot styles already defined in them, but you cannot modify
the plot styles or create new ones.

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.

**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.

**NOTE** If you insert an xref into your current drawing, all defined plot style tables are also inserted.

**Quick Reference**

**Commands**

PAGESETUP

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

**Use Color-Dependent Plot Style Tables**

By using color-dependent plot styles to control how objects are plotted, you ensure that all objects that share the same color are plotted the same way.

**NOTE** On the Mac, you can plot drawings with plot styles already defined in them, but you cannot modify the plot styles or create new ones.

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.
Use Predefined Color-Dependent Plot Style Tables
Several color-dependent plot style tables are installed in the Plot Styles folder, also known as the Plot Style Manager.

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>acad.ctb acadlt.ctb</td>
<td>Default plot style table</td>
</tr>
<tr>
<td>fillPatterns.ctb</td>
<td>Sets first 9 colors to use first 9 fill patterns, all others to use object's fill</td>
</tr>
<tr>
<td>grayscale.ctb</td>
<td>Converts all colors to grayscale when plotted</td>
</tr>
<tr>
<td>monochrome.ctb</td>
<td>Plots all colors as black</td>
</tr>
<tr>
<td>None</td>
<td>Applies no plot style table</td>
</tr>
<tr>
<td>screening 100%.ctb</td>
<td>Uses 100% ink for all colors</td>
</tr>
<tr>
<td>screening 75%.ctb</td>
<td>Uses 75% ink for all colors</td>
</tr>
<tr>
<td>screening 50%.ctb</td>
<td>Uses 50% ink for all colors</td>
</tr>
<tr>
<td>screening 25%.ctb</td>
<td>Uses 25% ink for all colors</td>
</tr>
</tbody>
</table>

**NOTE** You can assign a color-dependent plot style table to a layout only if the drawing has been set to use color-dependent plot style tables.

See also:
- Assign Plot Style Tables to Layouts on page 874

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.
NOTE  On the Mac, you can plot drawings with plot styles already defined in them, but you cannot modify the plot styles or create new ones.

An object whose plot style is set to BYLAYER inherits the plot style assigned to its layer.

Use the Properties Inspector palette to change an object's plot style and the Layers palette 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.

Quick Reference

Commands

LAYER

  Manages layers and layer properties.

PROPERTIES

  Controls properties of existing objects.

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 Print dialog box. The preview shows exactly how the drawing will look when plotted, including lineweights, fill patterns, and plot style options.

When you preview your drawing, the active toolbars and tool palettes are hidden and a temporary Preview toolbar is displayed that provides buttons to plot, pan, and zoom the drawing.
In the Plot and Page Setup dialog boxes, a thumbnail preview is also displayed, which shows the printable area and the position of the drawing on the page.

**Quick Reference**

**Commands**

PAN

Moves the view planar to the screen.

PLOT

Outputs a drawing to a printer or file.

PREVIEW

Displays the drawing as it will be printed.

ZOOM

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

**System Variables**

RASTERPREVIEW

Controls whether BMP preview images are saved with the drawing.

**Plot Files to Other Formats**

You can export or plot your drawings to other formats, including PDF and PostScript.

**Plot Adobe PDF Files**

You can create Adobe® Portable Document Format (PDF) files from drawings.

The Adobe® Portable Document Format (PDF) is a standard for electronic information exchange. PDF files can be easily distributed for viewing and printing in the Adobe Reader available from the Adobe web site without cost. Using PDF files, you can share drawings with virtually anyone.

Like DWF6 files, PDF files are generated in a vector-based format, for maintaining precision. Drawings that are converted to PDF can be easily distributed for viewing and printing in Adobe Reader, versions 7 or later.
Quick Reference

Commands

PLOT

Outputs a drawing to a printer or file.
Share Data Between Files
Attached xrefs are linked to, but not actually inserted in, another drawing. Therefore, with xrefs you can build drawings without significantly increasing the drawing file size.

**Overview of Referenced Drawings (Xrefs)**

By using referenced drawings, you can

- Coordinate your work with the work of others by referencing other drawings in your drawing to keep up with the changes being made by other designers. You can also assemble a master drawing from component drawings that may undergo changes as a project develops.

- Ensure that the most recent version of the referenced drawing is displayed. When you open your drawing, each referenced drawing is automatically reloaded, so it reflects the latest state of the referenced drawing file.

- Keep the names of layers, dimensioning styles, text styles, and other named elements in your drawing separate from those in referenced drawings.

- Merge (bind) attached referenced drawings permanently with your current drawing when the project is complete and ready to be archived.

**NOTE** Like a block reference, an xref appears in the current drawing as a single object. However, you cannot explode an xref without binding it first.
Quick Reference

Commands

BASE
Sets the insertion base point for the current drawing.

EXTERNALREFERENCES
Opens the Reference Manager palette.

EXTERNALREFERENCESCLOSE
Closes the Reference Manager palette.

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

XATTACH
Inserts DWG files as an external reference (xref).

XBIND
Binds one or more definitions of named objects in an xref to the current drawing.

XCLIP
Crops the display of a selected external reference or block reference to a specified boundary.

XREF
Starts the EXTERNALREFERENCES command.

System Variables

BINDTYPE
Controls how xref names are handled when binding xrefs or editing xrefs in place.

INSBASE
Stores the insertion base point set by BASE, which gets expressed as a UCS coordinate for the current space.

VISRETAIN
Controls the properties of xref-dependent layers.
XCLIPFRAME
Determines whether xref clipping boundaries are visible or plotted in the current drawing.

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

XLOADPATH
Creates a path for storing temporary copies of demand-loaded xref files.

Attach and Detach Referenced Drawings
You can perform several operations on referenced drawing files (xrefs).

Attach Drawing References (Xrefs)
You can insert any drawing file as an external reference or xref in the current drawing.

When you attach a drawing file as an xref, you link that referenced drawing to the current drawing. Any changes to the referenced drawing are displayed in the current drawing when it is opened or reloaded.

A drawing file can be attached as an xref to multiple drawings at the same time. Conversely, multiple drawings can be attached as referenced drawings to a single drawing.

Tools for Attaching Xrefs
You can use several methods to attach an xref:

■ On the menu bar, click Tools ➤ Palettes ➤ Reference Manager.
■ At the Command prompt, enter externalreferences
■ At the Command prompt, enter xattach.

The saved path used to locate the xref can be a full path, a relative (partially specified) path, or no path.
If an xref contains any variable block attributes, they are ignored.
Highlight External References in a Drawing

To find an external reference in a complex drawing, select an item in the Reference Manager palette to highlight all visible instances in the drawing. Conversely, select an external reference in the drawing to highlight its name in the Reference Manager palette.

Control the Properties of Referenced Layers

You can control the visibility, color, linetype, and other properties of an xref's layers and make these changes temporary or permanent. If the VISRETAIN system variable is set to 0, these changes apply only to the current drawing session. They are discarded when you end the drawing session, or when you reload or detach the xref.

You can also control the fade display of the DWG xref. The XDWGFADECTL system variable defines the fade percentage for all DWG xrefs.

Xref Clipping Boundaries

Drawings can include xrefs that are clipped. If you want to see the clipping boundary, you can turn on the XCLIPFRAME system variable.

Attachments from Educational Products

If you open, insert, or attach an xref from an Autodesk Educational Product, the drawings you plot contain the following banner: “PRODUCED BY AN AUTODESK EDUCATIONAL PRODUCT.”

See also:

- Nest and Overlay Referenced Drawings on page 886
- Clip External References and Blocks on page 894
- Set Paths to Referenced Drawings on page 888
- Set Interface Options on page 43

To attach an xref

1. On the menu bar, click Tools ➤ Palettes ➤ Reference Manager.
2. In the Reference Manager, click the Attach Reference button.
3. In the Select Reference File dialog box, locate and click the file to be referenced. Click Open.
4 In the Attach External Reference dialog box, select any desired options and then click OK.

5 If necessary, specify the location in the drawing and any other options.

**Quick Reference**

**Commands**

EXTERNALREFERENCES

Opens the Reference Manager palette.

EXTERNALREFERENCESCLOSE

Closes the Reference Manager palette.

XATTACH

Inserts DWG files as an external reference (xref).

XREF

Starts the EXTERNALREFERENCES command.

**System Variables**

ERHIGHLIGHT

Controls whether reference names or reference objects are highlighted when their counterparts are selected in the External References palette or in the drawing window.

VISRETAIN

Controls the properties of xref-dependent layers.

XDWGFADECTL

Controls the dimming for all DWG xref objects.

XLOADCTL

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

XLOADPATH

Creates a path for storing temporary copies of demand-loaded xref files.
Nest and Overlay Referenced Drawings

Attached DWG references (xrefs) can be nested: that is, you can attach an xref that contains another xref.

Xrefs can be nested within other xrefs: that is, you can attach an xref that contains another xref. You can attach as many copies of an xref as you want, and each copy can have a different position, scale, and rotation.

In the following illustration, master.dwg references a.dwg and b.dwg. Drawing a.dwg references c.dwg. In master.dwg, c.dwg is a nested xref.

You can also overlay an xref on your drawing. Unlike an attached xref, an overlaid xref is not included when the drawing is itself attached or overlaid as an xref to another drawing. Overlaid xrefs are designed for data sharing in a network environment. By overlaying an xref, you can see how your drawing relates to the drawings of other groups without changing your drawing by attaching an xref.

In the following illustration, several people are working on drawings referenced by master.dwg. The person working on a.dwg needs to see the work being completed by the person working on b.dwg, but does not want to xref b.dwg because it would then appear twice in master.dwg. Instead, the person overlays b.dwg, which is not included when a.dwg is referenced by master.dwg.
NOTE When using the parametric drawing feature, you can only constrain objects in the drawing to the insertion point of an Xref, and not its nested objects.

Relative Saved Paths and Nested Xrefs

The saved path for an xref can be a full path, a relative (partially specified) path, or no path. For a nested xref, a relative path always references the location of its immediate host and not necessarily the currently open drawing.

See also:

Quick Reference

Commands

EXTERNALREFERENCES

Opens the Reference Manager palette.

System Variables

XREFTYPE

Controls the default reference type when attaching or overlaying an external reference.
Set Paths to Referenced Drawings

You can view and edit the file name and path used when locating a particular drawing reference (xref). Use this option if the referenced file has been moved to a different folder or renamed since it was first attached.

You can choose from three types of folder path information to save with an attached reference: a full path, a relative path, and no path.

Specify a Full (Absolute) Path

A full path is a fully specified hierarchy of folders that locates the file reference. For example, a fully specified path to a different volume will look something like this:

```
smb://hostname/directorypath/resource
```

Instead of smb:, you could use afp:, ftp:, or other protocol.

This is the most specific but least flexible option.

Specify a Relative Path

Relative paths are partially specified folder paths that assume the current folder of the host drawing. This is the most flexible option, and enables you to move a set of drawings from one folder to a different one that contains the same folder structure.

If the file that is being referenced is located on a network server, the relative path option is not available.

The conventions for specifying a relative folder path are as follows:

```
/  Look in the root folder of the host drawing's drive
path  From the folder of the host drawing, follow the specified path
/path  From the root folder, follow the specified path
./path  From the folder of the host drawing, follow the specified path
../path  From the folder of the host drawing, move up one folder level and follow the specified path
../../path  From the folder of the host drawing, move up two folder levels and follow the specified path
```

NOTE If a drawing that contains referenced files is moved or saved to a different path, 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
- Search paths defined in the Project Files Search Path on the Application tab in the Application Preferences dialog box
- Search paths defined in the Support File Search Paths on the Application tab in the Application Preferences dialog box

Specifying the No Path option is useful when moving a set of drawings to a different folder hierarchy or to an unknown folder hierarchy.

Know when a Referenced Drawing has been Relocated

If the drawing you are working on contains an xref that has been moved to a different folder, a message is displayed at the site of the xref when you load the drawing. The message indicates that the xref cannot be loaded using the old path. When you specify the new path, the xref is reloaded into your drawing.

Quick Reference

Commands
XREF

Starts the EXTERNALREFERENCES command.

Detach Referenced Drawings

To completely remove DWG references (xrefs) from your drawing, you need to detach them rather than erase them.
Erasing xrefs does not remove, for example, layer definitions associated with those xrefs. Using the Detach option removes the xrefs and all associated information.

To detach an xref

1. On the menu bar, click Tools ➤ Palettes ➤ Reference Manager.
2. In the Reference Manager, click a DWG reference.
3. Right-click the selected DWG reference and select Detach from the shortcut menu. Alternatively, you can click the Detach button in the top row of buttons in the Reference Manager.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the Reference Manager palette.

Update and Archive Referenced Drawings

You can update referenced drawings (xrefs) to make sure that they are current, and you can choose how xrefs are treated when a drawing is archived.

Update Referenced Drawing Attachments

When you open a drawing, all drawing references (xrefs) update automatically. You can also update xrefs whenever you want to ensure that the most current versions are displayed in your drawing.

When you open a drawing, all xrefs update automatically. Use the Refresh Content (Reload) option from the Reference Manager to update xrefs whenever you want to ensure that the most current versions are displayed in your drawing.
Whenever you modify and save an externally referenced drawing in a network environment, other people can access your changes immediately by reloading the xrefs in their open drawings.

**Update Xrefs with Demand Loading Turned On**

If demand loading is turned on when you load or reload an xref

- With the XLOADCTL system variable set to 1, the referenced drawing is kept open and locked. No one else can modify the referenced drawing.
- With XLOADCTL set to 2, a temporary copy of the most recently saved version of the referenced file is opened and locked. Others can open and modify the referenced drawing.

For information about demand loading, see Increase Performance with Large Referenced Drawings on page 904.

**To update an attached xref**

1. On the menu bar, click Tools ➤ Palettes ➤ Reference Manager.
2. In the Reference Manager, click a DWG reference.
3. Right-click the selected DWG reference and select Reload from the shortcut menu. Alternatively, you can click the Refresh Content button in the top row of buttons in the Reference Manager.

**NOTE** If the drawing you selected has been changed since you opened your drawing, the xref is reloaded.
Quick Reference

Commands
EXTERNALREFERENCES
  Opens the Reference Manager palette.
EXTERNALREFERENCESCLOSE
  Closes the Reference Manager palette.

System Variables
XLOADCTL
  Turns xref demand-loading on and off, and controls whether it opens the referenced drawing or a copy.
XLOADPATH
  Creates a path for storing temporary copies of demand-loaded xref files.

Archive Drawings That Contain Referenced Drawings (Bind)
When you archive final drawings that contain xrefs, you can choose how you store the xrefs in the drawings.

When you archive final drawings that contain xrefs, you have two choices:

■ Store the xref drawings along with the final drawing
■ Bind the xref drawings to the final drawing

Storing an xref drawing along with the final drawing requires that the drawings always remain together. Any change to the referenced drawing will continue to be reflected in the final drawing.

To prevent unintentional updating of archived drawings by later changes to referenced drawings, bind the xrefs to the final drawing.

Binding an xref to a drawing makes the xref a permanent part of the drawing and no longer an externally referenced file. You can bind the entire database of the xref drawing, including all its xref-dependent named objects (blocks, dimension styles, layers, linetypes, and text styles), by using the XREF Bind option. For more information, see Resolve Name Conflicts in External References on page 900.
Binding xrefs to a drawing is also an easy way to send a drawing to reviewers. Rather than sending a master drawing plus each of the drawings it references, you can use the Bind option to merge the xrefs into the master drawing.

**NOTE** You cannot bind xrefs that contain proxy objects. For more information, see [Work with Custom and Proxy Objects](#) on page 937.

**To bind an xref to the current drawing**

1. On the menu bar, click Tools ➤ Palettes ➤ Reference Manager.
2. In the Reference Manager, click a DWG reference.
3. Right-click the selected DWG reference and select Bind from the shortcut menu.

   The objects in the xref are converted into a block reference. Named object definitions are added to the current drawing with a prefix of `blockname $n$`, where $n$ is a number starting at 0.

**Quick Reference**

**Commands**

**EXTERNALREFERENCES**

Opens the Reference Manager palette.

**XBIND**

Binds one or more definitions of named objects in an xref to the current drawing.

**System Variables**

**BINDTYPE**

Controls how xref names are handled when binding xrefs or editing xrefs in place.

**XEDIT**

Controls whether the current drawing can be edited in-place when being referenced by another drawing.
Clip External References and Blocks

You can specify clipping boundaries to display a limited portion of an external reference drawing or block reference.

You can clip external references such as DGN, DWF, IMAGE, PDF underlays, or block references. With a clipping boundary, you can determine the portions of an external reference or block reference that you want to display by hiding the redundant parts of the reference inside or outside the boundary.

The clipping boundary can be a polyline, rectangle, or a polygon with vertices within the boundaries of the image. You can change the boundary of a clipped image. When you clip a boundary, the objects in the external reference or block are not altered; only their display is changed.

With the XCLIP and IMAGECLIP commands, you can control the following viewing options:

Control the visibility of the clipped area of the external reference or block reference. When clipping is turned off, the boundary is not displayed and the entire external reference or block is visible, provided that the objects are on layers that are turned on and thawed.

Clipping results can be turned on or off using the clipping commands. This controls whether the clipped area is hidden or displayed.

Control the visibility of clipping boundaries.
Invert the area to be hidden, inside or outside the clipping boundary. When you want the hidden parts of the clipped reference displayed or vice versa, use the grips to alter the display of the external reference or blocks. With grips located at the midpoint on the first edge of the clipping boundary, you can invert the display of the clipped reference inside or outside the boundary.

The grips are visible and can be used when the clipping system variable is turned on, the reference is selected, and clipped.

**Editing Options**

After an external reference or block reference has been clipped, it can be moved, copied, or rotated just like an unclipped external reference or block reference. The clipping boundary moves with the reference. If an xref contains nested clipped xrefs, they appear clipped in the drawing. If the parent xref is clipped, the nested xrefs are also clipped.

**Resize Clipping Boundaries**

If you want to change the shape or size of a clipping boundary for external references and block references, you can use grips to edit the vertices just as you edit any object with grips.

In case of rectangular grip editing, you can maintain the closed four-sided rectangle or square shape of the rectangular clipping boundary because two
vertices of the same side of the rectangular clipping boundary are edited together.

**NOTE** With Clipping boundaries, you cannot display self-intersecting polygonal boundaries. An error message is displayed and the boundary reverts to the last boundary.

**Limitations for Clipping Boundaries**

When clipping an referenced drawing or block the following limitations apply:

- A clipping boundary can be specified anywhere in 3D space, but it is always applied planar to the current UCS.
- If a polyline is selected, the clipping boundary is applied in the plane of that polyline.
- Images in external references or blocks are always clipped within the rectangular extents of the reference. When you apply polygonal clipping to images in externally referenced drawings, the clipping boundary is applied to the rectangular extents of the polygonal boundary, rather than to the polygon itself.

See also:

- [Clip Raster Images](#) on page 914

**Quick Reference**

**Commands**

**IMAGECLIP**

Crops the display of a selected image to a specified boundary.

**XCLIP**

Crops the display of a selected external reference or block reference to a specified boundary.

**System Variables**

**FRAMESELECTION**

Controls whether the frame of an image, underlay, or clipped xref can be selected.
IMAGEFRAME

Controls whether image frames are displayed and plotted.

XCLIPFRAME

Determines whether xref clipping boundaries are visible or plotted in the current drawing.

Edit Referenced Drawings

Referenced drawings can be edited by opening them directly, or you can edit the xref in place from within the current drawing. You can edit a block definition directly from any selected block reference.

Edit a Referenced Drawing in a Separate Window

While the simplest and most direct method for editing xrefs is to open the source file for the referenced drawing, there is an alternative that can be more convenient.

If you need to edit the model space objects in an xref, you can access the xref or a nested xref directly from the Reference Manager or with the XOPEN command. Select the xref, and then using the shortcut menu in the Reference Manager, open the xref's source file. After you save the edits, close the drawing. In your original drawing, click the Refresh Content button in the Reference Manager, and resume working.

NOTE Make sure you know whether the referenced drawing is also referenced by other drawings, and the changes you make are appropriate in other instances.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the Reference Manager palette.

XOPEN

Opens a selected drawing reference (xref) in a new window.
Resolve Referenced Drawing Errors

If a referenced drawing cannot be loaded when you open a drawing, an error message is displayed.

Resolve Missing External References

If a referenced drawing cannot be located when you open a drawing, several options available to you.

The program stores the folder path of the referenced drawing. Each time you open or plot the drawing, or use the Reload option in the Reference Manager to update the xref, the program checks the folder 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 locate or reload the xref, and it displays an error message that displays the folder path and name of the missing drawing file.

In the drawing, at each insertion of the missing xref, the program displays text that displays the folder path of the missing xref. You can use the XREF Path option to update or correct the path.

Along with error messages being displayed at the Command prompt, a task dialog box might be displayed that allows you to ignore all missing xrefs or update their folder locations. You can use the Reference Manager palette to update the locations of the unresolved references.

To avoid these errors make sure that when you transfer or distribute drawing files that have xrefs attached, you also include all the referenced files.

Change Nested Xref Paths

When a drawing is opened and a nested xref is loaded, the program attempts to find the xref in the original xref path first. If the xref is not found, the following search is initiated in the order shown:

- Current folder of the host drawing
- Search paths defined in the Project Files Search Path on the Application tab in the Application Preferences dialog box
- Search paths defined in the Support File Search Paths on the Application tab in the Application Preferences dialog box
This search order helps ensure that revisions made to the xref are reflected in the current drawing, and also makes it possible for the xref to be found if its folder path has changed.

See also:

■ Update Referenced Drawing Attachments on page 890

Quick Reference

Commands

EXTERNALREFERENCES

Opens the Reference Manager palette.

Resolve Circular External References

If a referenced drawing contains a sequence of nested references that refers back to itself, an error message is displayed.

A drawing that contains a sequence of nested references that refers back to itself is considered a circular reference. For example, if drawing A attaches drawing B, which attaches drawing C, which attaches drawing A, the reference sequence A>B>C>A is a circular reference.

If the program detects a circular reference while attaching an xref, a warning is displayed asking you if you want to continue. If you respond with yes, the program reads in the xref and any nested xrefs to the point where it detects the circularity. If you respond with no, the process is halted and the xref is not attached.

If a circular reference is encountered while loading a drawing, an error message is displayed and the circular reference for the current session is broken. For example, if you have the circular reference A>B>C>A, and you open a.dwg, the program detects and breaks the circularity between c.dwg and a.dwg. The following error message is displayed:

Breaking circular reference from C to current drawing.
Quick Reference

Commands

EXTERNALREFERENCES

Opens the Reference Manager palette.

Resolve Name Conflicts in External References

When you attach an xref, the names of its blocks, dimension styles, layers, linetypes, and text styles are differentiated from those in the current drawing.

A typical xref definition includes objects, such as lines or arcs. It also includes xref-dependent definitions of blocks, dimension styles, layers, linetypes, and text styles. When you attach an xref, the program differentiates the names of these xref-dependent named objects from those in the current drawing by preceding their names with the name of the referenced drawing and a vertical bar character (|). For example, in the Layer Properties Manager, the xref-dependent named object that is a layer named STEEL in a referenced drawing called stair.dwg is listed as STAIR|STEEL.

When you attach an xref, the definitions of its dependent named objects are not added to your drawing permanently. Instead, these definitions are loaded from the referenced drawing file each time you reload it.

Bind Xref-Dependent Definitions

An xref-dependent named object’s definition can change if the referenced drawing file is modified. For example, a layer name from a referenced drawing can change if the referenced drawing is modified. The layer name can even disappear if it is purged from the referenced drawing. This is why the program does not allow you to use an xref-dependent layer or other named object directly. For example, you cannot insert an xref-dependent block or make an xref-dependent layer the current layer and begin creating new objects on it.

To avoid the restrictions on xref-dependent named objects, you can bind them to your current drawing. Binding makes the xref-dependent named objects that you select become a permanent part of your current drawing.

When xref-dependent named objects are merged into a drawing through binding, you can use them the same way you use the drawing’s own named objects. After you bind an xref-dependent named object, the vertical bar character (|) is removed from the name and replaced with two dollar signs ($$) separated by a number (usually zero): for example, the referenced layer,
STAIR\$STEEL, becomes STAIR\$STEEL. You can then use the RENAME command to change STAIR\$STEEL to STEEL.

If you specify a layer whose associated linetype is not CONTINUOUS, the referenced linetype is also bound. If you apply XBIND to a block, all named objects that are referenced by the objects in the block are also bound. If the block contains a reference to an xref, that xref and all of its dependent definitions are bound.

**Quick Reference**

**Commands**

RENAME

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

XBIND

Binds one or more definitions of named objects in an xref to the current drawing.

**System Variables**

BINDTYPE

Controls how xref names are handled when binding xrefs or editing xrefs in place.

**Track External Reference Operations (Log File)**

You can maintain a record of actions while attaching, detaching, and reloading xrefs, and while loading a drawing containing xrefs.

This log is maintained only if the XREFCTL system variable is set to 1. The default setting is 0.

The log file is an ordinary ASCII text file with the same name as the current drawing and the file extension .xlg. If you load a drawing with the file name sample.dwg, for example, the program searches for a log file named sample.xlg in the current folder. If the file does not exist, a new file is created with that name.

Once a log file has been created for a drawing, the program continues to append information to it. The program writes a title section to the log file each time the file is opened. If the log file becomes too large, you can delete it.
Example: A Sample Title Section from an Xref Log File
This title section contains the name of the current drawing, the date and time, and the operation being performed.

Drawing: detail  
Date/Time: 09/28/99 10:45:20  
Operation: Attach Xref

During a detaching or reloading operation, the program includes the nesting level of all affected xrefs immediately following the title section. To see a reference tree for a set of xrefs in your current drawing, use Detach or Reload and check the resulting entries in the log file.

Example: A Sample Log File Entry Showing Nested Xrefs
In the following example, the xref ENTRY_DR contains two nested xrefs: HARDWARE and PANELS. The xrefs HARDWARE and PANELS also each contain two xrefs.

Drawing: detail  
Date/Time: 10/05/99 15:47:39  
Operation: Reload Xref

Reference tree for ENTRY_DR:  
ENTRY_DR Xref  
  -HARDWARE Xref  
    --LOCKSET Xref  
    --HINGES Xref  
  -PANELS Xref  
    --UPPER Xref  
    --LOWER Xref

The program writes an entry in the log file for each xref-dependent named object temporarily added to the current drawing and for any errors that occur. Most error messages are written both to the screen and to the log file.

Example: A Sample Log File That Shows the Results of Attaching an Xref
The following example shows a partial listing of the log file entries generated when the external reference STAIR is attached to the working drawing test.dwg.
The log file lists the definition (symbol) table affected and the name of the
definition added, along with a status message.

---

Drawing: test
Date/Time: 12/18/99 14:06:34
Operation: Attach Xref
---

Attach Xref STAIR: \ACAD\DWGS\STAIR.dwg
Searching in ACAD search path
Update block symbol table:
  Appending symbol: STAIR|BOLT
  Appending symbol: STAIR|BOLT-HALF
...
  block update complete.
Update Ltype symbol table:
  Appending symbol: STAIR|DASHED
  Appending symbol: STAIR|CENTER
  Appending symbol: STAIR|PHANTOM
  Ltype update complete.
Update Layer symbol table:
  Appending symbol: STAIR|STEEL-HIDDEN
  Appending symbol: STAIR|OAK
...
  Layer update complete.
STAIR loaded.

To use the xref log file
1  At the Command prompt, enter xrefctl.
2  Enter 1 to turn logging on or 0 to turn logging off.
3  Press Enter.
   Logging is off by default.

Quick Reference

Commands
XREF
  Starts the EXTERNALREFERENCES command.
System Variables

XREFCTL

Controls whether external reference log (XLG) files are created.

Increase Performance with Large Referenced Drawings

There are several features that can improve performance when dealing with large referenced drawings.

Overview of Demand Loading

The program uses demand loading and saving drawings with internal indexes to increase performance with large referenced drawings that have been clipped, or that have many objects on frozen layers. With demand loading, only the data from the reference drawing that is necessary to regenerate the current drawing is loaded into memory. In other words, referenced data is read in “on demand.”

Demand loading works in conjunction with the XLOADCTL and XLOADPATH system variables.

Quick Reference

Commands

XREF

Starts the EXTERNALREFERENCES command.

System Variables

XLOADCTL

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

XLOADPATH

Creates a path for storing temporary copies of demand-loaded xref files.
Unload Xrefs in Large Drawings

When a referenced drawing (xref) is unloaded from the current drawing, the drawing opens much faster and uses less memory.

The xref definition is unloaded from the drawing file, but the internal pointer to the referenced drawing remains. The xref is not displayed, and non-graphical object information does not appear in the drawing. However, you can restore all the information by reloading the xref. If XLOADCTL (demand loading) is set to 1, unloading the drawing unlocks the original file.

You should unload a reference file if it is not needed in the current drawing session but may be used later for plotting. You can maintain a working list of unloaded xrefs in the drawing file that you can load as needed.

Quick Reference

Commands

EXTERNALREFERENCES
Opens the Reference Manager palette.

System Variables

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

Set Paths for Temporary Xref File Copies

When you turn on demand loading with copy, you can control where copies of externally referenced drawings are to be placed.

When you turn on demand loading with copy, the XLOADPATH system variable can be used to indicate the path where copies of externally referenced drawings are to be placed. The path you specify remains in effect for all drawing sessions until you indicate a different path. If no value for XLOADPATH is specified, the temporary file copies are placed in the standard folder for temporary files.

If you find that referencing drawings over a network is slow, it is recommended that you set XLOADPATH to reference a local folder, and set XLOADCTL to 2 so that the externally referenced files are demand loaded from your local machine. Conversely, to minimize the number of temporary files created by
multiple users referencing the same drawing, those users can set XLOADPATH to point to a common folder. In this manner, multiple sessions of the program can share the same temporary copies of reference drawings.

You can set XLOADPATH in the Application Preferences dialog box, Application tab, Temporary External Reference File Location, and indicate the folder path where copies of externally referenced files are to be placed.

Quick Reference

Commands

EXTERNALREFERENCES

- Opens the Reference Manager palette.

OPTIONS

- Customizes the program settings.

System Variables

XLOADPATH

- Creates a path for storing temporary copies of demand-loaded xref files.
You can work with many different types of files, including files created with other applications and files created in earlier releases of the program. You can also specify search paths for drawing and support files.

Import Other File Formats

You can import files, other than DWG files, that were created with other applications into your drawings.

Import ACIS SAT Files

You can import geometry objects stored in SAT (ASCII) files using the ASCISIN command.

ACISIN converts the model to a body object or to 3D solids and regions if the body is a true solid or a true region.

Quick Reference

Commands

ACISIN
Imports an ACIS (SAT) file and creates 3D solid, body, or region objects.

IMPORT
Imports files of different formats into the current drawing.
Convert DXF and DXB Files to DWG Format

DXF and DXB files are two types of drawing interchange files used to transfer data between various applications.

A DXF (drawing interchange format) file is either a binary or an ASCII representation of a drawing file. It is often used to share drawing data between other CAD programs.

A DXB (drawing interchange binary) file is a specially coded binary version of a DXF file used for plotting, and can be used to “flatten” 3D wireframe drawings into 2D vectors.

You can convert a DXF or DXB file to DWG format by opening the file and saving it in DWG format. You can then work with the resulting drawing file as you would with any other drawing file.

Quick Reference

Commands

DXBIN

Imports an AutoCAD DXB (drawing interchange binary) file.

OPEN

Opens an existing drawing file.

Attach Raster Image Files

You can view and manipulate raster images and associated file paths in drawings.

You can add raster images to your vector-based drawings, and then view and plot the resulting file. There are a number of reasons for combining raster images with vector files, including scanning documents, faxes, or microfilm drawings; using aerial and satellite photographs; using digital photographs; creating effects such as watermarks and logos; and adding computer-rendered images.

Overview of Raster Images

Raster images consist of a rectangular grid of small squares or dots known as pixels. For example, a photograph of a house is made up of a series of pixels.
colorized to represent the appearance of a house. A raster image references the pixels in a specific grid.

Raster images, like many other drawing objects, can be copied, moved, or clipped. You can modify an image with grip modes, adjust an image for contrast, clip the image with a rectangle or polygon, or use an image as a cutting edge for a trim.

The image file formats supported by the program include the most common formats used in major technical imaging application areas: computer graphics, document management, engineering, mapping, and geographic information systems (GIS). Images can be bitonal, 8-bit gray, 8-bit color, or 24-bit color. Images with 16-bit color depth are not supported starting with AutoCAD 2011 for Mac.

Several image file formats support images with transparent pixels. When image transparency is set to on, the program recognizes those transparent pixels and allows graphics in the drawing area to “show through” those pixels. (In bitonal images, background pixels are treated as transparent.) Transparent images can be gray-scale or color.

**NOTE** Although the file name extension is listed in the following table, the file format is determined from the file contents, not from the file extension.

### Supported image file formats

<table>
<thead>
<tr>
<th>Type</th>
<th>Description and versions</th>
<th>File extension</th>
</tr>
</thead>
<tbody>
<tr>
<td>BMP</td>
<td>Windows and OS/2 bitmap format</td>
<td>.bmp, .dib, .rle</td>
</tr>
<tr>
<td>CALS-I</td>
<td>Mil-R-Raster I</td>
<td>.gp4, .mil, .rst, .cg4, .cal</td>
</tr>
<tr>
<td>FLIC</td>
<td>FLIC Autodesk Animator Animation</td>
<td>.flic, .fli</td>
</tr>
</tbody>
</table>
Supported image file formats

<table>
<thead>
<tr>
<th>Type</th>
<th>Description and versions</th>
<th>File extension</th>
</tr>
</thead>
<tbody>
<tr>
<td>GeoSPOT</td>
<td>GeoSPOT (BIL files must be accompanied with HDR and PAL files with correlation data, in the same directory)</td>
<td>.bil</td>
</tr>
<tr>
<td>IG4</td>
<td>Image Systems Group 4</td>
<td>.ig4</td>
</tr>
<tr>
<td>JFIF or JPEG</td>
<td>Joint Photographics Expert Group</td>
<td>.jpg or .jpeg</td>
</tr>
<tr>
<td>PCX</td>
<td>Picture PC Paintbrush Picture</td>
<td>.pcx</td>
</tr>
<tr>
<td>PICT</td>
<td>Picture Macintosh Picture</td>
<td>.pct</td>
</tr>
<tr>
<td>PNG</td>
<td>Portable Network Graphic</td>
<td>.png</td>
</tr>
<tr>
<td>RLC</td>
<td>Run-Length Compressed</td>
<td>.rlc</td>
</tr>
<tr>
<td>TARGA</td>
<td>True Vision Raster-Based Data Format</td>
<td>.tga</td>
</tr>
<tr>
<td>TIFF</td>
<td>Tagged Image File Format</td>
<td>.tif or .tiff</td>
</tr>
</tbody>
</table>

Quick Reference

Commands

IMAGE
Displays the Reference Manager palette.

IMAGEATTACH
Inserts a reference to an image file.

Attach, Scale, and Detach Raster Images
You can add or remove references to raster images within drawing files, or you can change their relative size.
Attach Raster Images

You can attach a reference to a raster image file to a drawing file using a linked image path. The image file can be accessed from the Internet.

Images can be referenced and placed in drawing files, but like external references (xrefs), they are not actually part of the drawing file. The image is linked to the drawing file through a path name. Linked image paths can be changed or removed at any time.

Once you've attached an image, you can reattach it multiple times, treating it as if it were a block. Each insertion has its own clip boundary and its own settings for brightness, contrast, fade, and transparency.

NOTE AutoCAD 2000, AutoCAD LT 2000, and later releases do not support LZW-compressed TIFF files, with the exception of English language versions sold in the US and Canada. If you have TIFF files that were created using LZW compression and want to insert them into a drawing, you must resave the TIFF files with LZW compression disabled.

For information on identifying referenced images, see Highlight External References in a Drawing in Attach and Detach Referenced Drawings on page 883

Access Raster Images Using the Internet

Designers and manufacturers store images of their designs or products on the Internet. You can easily access image files from the Internet. URL image file names are stored in the drawing.

Accessing images from the Internet saves time and provides for rapid distribution of designs. For example, an architect who needs to show a client what custom cabinets will look like has the manufacturer create a rendered image of the cabinets, post it to a website, and then attach the image to the drawing file as a URL; any design changes can be updated immediately. For more information, see Reference Other Drawing Files on page 881.

Quick Reference

Commands

IMAGE

Displays the Reference Manager palette.
Scale Raster Images

You can control the size of a raster image in a drawing to match the scale of the drawing.

You can specify the raster image scale factor when you attach the image so that the scale of the geometry in the image matches the scale of the geometry in the drawing. The default image scale factor is 1, and the default unit for all images is “Unitless.” The image file can contain resolution information defining the dots per inch (DPI), relating to how the image was scanned.

If an image has resolution information, the program combines this information with the scale factor and the unit of measurement of the drawing to scale the image in your drawing. For example, if your raster image is a scanned blueprint on which the scale is 1 inch equals 50 feet, or 1:600, and your drawing is set up so that 1 unit represents 1 inch, then in the Image dialog box under Scale, select Specify On-Screen. To scale the image, you clear Specify On-Screen, and then enter 600 in Scale. The image is then attached at a scale that brings the geometry in the image into alignment with the geometry in the drawing.

If no resolution information is defined with the attached image file, the width of the raster image is set to one unit. Thus, when the image file is attached, the image width in units is equal to the raster image scale factor.

Quick Reference

Commands

IMAGE
   Displays the Reference Manager palette.

IMAGEATTACH
   Inserts a reference to an image file.

Detach Raster Images

You can detach the reference to an image file in a drawing.
You can detach images that are no longer needed in a drawing. When you detach an image, all instances of the image are removed from the drawing, the image definition is purged, and the link to the image is removed. The image file itself is not affected.

**NOTE** Erasing an individual instance of an image is not the same as detaching an image. An image must be detached to remove the link from your drawing to the image file.

### Quick Reference

**Commands**

EXTERNALREFERENCES

Opens the Reference Manager palette.

### Modify Raster Images and Image Boundaries

You can control the clipping boundaries and image display properties of a raster image.

### Show and Hide Raster Image Boundaries

You can control whether the clipping boundaries of a raster image are displayed or hidden in a drawing.

You can hide image boundaries. Hiding the image boundary prevents the boundary from being plotted or displayed. Also, hiding the image boundary prevents you from selecting the image with the pointing device, ensuring that the image cannot accidentally be moved or modified. However, images can still be selected if they are not on a locked layer, for example, if the image is part of a named selection set made with the All option. When image boundaries are hidden, clipped images are still displayed to their specified boundary limits; only the boundary is affected. Showing and hiding image boundaries affects all images attached to your drawing.
NOTE When an image frame is turned off, you cannot select images using the Pick or Window options of SELECT.

Quick Reference

Commands

IMAGEFRAME

Controls whether image frames are displayed and plotted.

System Variables

FRAMESELECTION

Controls whether the frame of an image, underlay, or clipped xref can be selected.

Clip Raster Images

You can clip and display specific portions of a raster image in a drawing with a clipping boundary.

With a clipping boundary, only the parts of the image that you want visible are displayed. You can define the part of an image that you want to display and plot by clipping the image with IMAGECLIP. The clipping boundary can be a polyline, rectangle, or a polygon with vertices within the boundaries of the image. You can change the boundary of a clipped image. You can also delete the clipped boundary of an image. When you delete a clipping boundary, the original image is displayed.

You can invert the area to be hidden, inside or outside the clipping boundary. With grips located at the midpoint on the first edge of the clipping boundary,
you can invert the display of the clipped reference inside or outside the boundary.

With IMAGEFRAME system variable, you can control the visibility of the clipping boundary.

See also:
- Clip External References and Blocks on page 894

Quick Reference

Commands

IMAGECLIP
Crops the display of a selected image to a specified boundary.

PROPERTIES
Controls properties of existing objects.

System Variables

IMAGEFRAME
Controls whether image frames are displayed and plotted.

Change Raster Image Brightness, Contrast, and Fade

You can change several display properties of raster images in a drawing for easier viewing or special effects.

You can adjust brightness, contrast, and fade for the display of an image as well as for plotted output without affecting the original raster image file and
without affecting other instances of the image in the drawing. Adjust brightness to darken or lighten an image. Adjust contrast to make poor-quality images easier to read. Adjust fade to make drawing geometry easier to see over images and to create a watermark effect in your plotted output.

Bitonal images cannot be adjusted for brightness, contrast, or fade. Images fade to the current screen background when displayed, and they fade to white when plotted.

Quick Reference

Commands

IMAGEADJUST

Controls the image display of the brightness, contrast, and fade values of images.

PROPERTIES

Controls properties of existing objects.

Modify Color and Transparency for Bitonal Raster Images

Bitonal images are images that consist only of a foreground color and a background color. You can change the foreground color and turn the transparency of the background color on and off.

Bitonal raster images are images consisting only of a foreground color and a background color. When you attach a bitonal image, the foreground pixels in the image inherit the current settings for color. In addition to the modifications you make to any attached image, you can modify bitonal images by changing the foreground color and by turning on and off the transparency of the background.

NOTE  Bitonal images and their boundaries are always the same color.

Quick Reference

Commands

PROPERTIES

Controls properties of existing objects.
TRANSPARENCY

Controls whether background pixels in an image are transparent or opaque.

Manage Raster Images

You can view and manipulate raster images and change paths to image files using the External References palette.

View Raster Image Information

You can view file-specific information about the raster images that are attached to a drawing. You can also load and unload the images and perform other operations using the External References palette.

In the External References palette, you can view image information either as a list or as a tree in the File References Pane. To control how the information is displayed in the External References palette, click the List View or Tree View button in the upper-right corner. The list view displays the name of each image in the drawing, its loading status, size, date last modified, and search path. The tree view lists the images in a hierarchy that shows their nesting level within referenced drawings and blocks. The status, size, and other information is displayed below in the Details pane.

In either view, you can display information about an image; attach or detach the image; unload or reload the image; and browse for and save a new search path.

Use the List View

The list view displays the images and any other external references attached to the current drawing, but it does not specify the number of instances. It is the default view. You can sort the external references by categories by clicking the column headings. Change the width of a column by dragging its border to the right or left.

The following information is displayed in the list view:

- Name of the image or selected external reference
- Status (loaded, unloaded, or not found)
- File size
- File type
- Date and time file was last saved
- Name of the saved path

If an image cannot be found, its status is listed as Not Found. A Not Found image is displayed as an image boundary in the drawing even if the IMAGEFRAME system variable is set to off. If the image is unreferenced, no instances are attached for the image. If the image is not loaded, its status is Unloaded. Images with a status of Unloaded or Not Found are not displayed in the drawing.

**Use the Tree View**

The top level of the tree view lists referenced files in the order that they were attached. In most cases an image file is linked directly to the drawing and listed at the top level. However, if a DWG file reference or a block contains a linked image, additional levels are displayed.

**View Image File Details**

In the lower panes of the External References palette, you can preview a selected image or view image file details, including
- Image name
- Saved path
- Active path (where the image is found)
- File creation date
- File size
- File type
- Color
- Color depth
- Image size (pixel width and height, resolution and default size)

**View Image Information in the Text Window**

You can view image information at the Command prompt. Command prompt image information includes image name, image path, the number of definitions, and the number of instances of the image attached to the drawing.
Assign Descriptive Names to Raster Images

When the name of a raster image file is not sufficient to identify an image, you can add a descriptive name using the External Reference palette.

Image names are not necessarily the same as image file names. When you attach an image to a drawing, the program uses the file name without the file extension as the image name. Image names are stored in a symbol table; thus you can change the image name without affecting the name of the file. Up to 255 characters are accepted for image file names. In addition to letters and numbers, names can have spaces and any special characters not used by Microsoft® Windows® or AutoCAD for Mac for other purposes.

If you attach and place images with the same name but from two different directories, numbers are appended to the image names.

Quick Reference

Commands

EXTERNALREFERENCES

Opens the Reference Manager palette.

IMAGEATTACH

Inserts a reference to an image file.

Change File Paths of Raster Images

With the External References palette, you can change the file path to a referenced raster image file or search for a referenced image when it is reported as not found.

When you open a drawing with an attached image, the path of the selected image is displayed in the Reference Manager in the Details Pane under Found At. The path displayed is the actual path where the image file was found. The
path where the image file was originally attached is displayed under Saved Path.

To locate the image file, the program searches the following paths and folders in the following order:

■ Path specified when the image was attached
■ Folder containing the current drawing file
■ Project search paths specified on the Application tab in the Application Preferences dialog box
■ Support search paths specified on the Application tab in the Application Preferences dialog box

You can remove the path from the file name or specify a relative path by editing the path in the Found At properties and then clicking OK in the Select Image File dialog box.

For more information about using full paths, relative paths, and project names, see Set Paths to Referenced Drawings on page 888.

Quick Reference

Commands
EXTERNALREFERENCES
  Opens the Reference Manager palette.
OPTIONS
  Customizes the program settings.

Tune Raster Image Performance

You can reduce the demands on system performance when manipulating large or many small raster images.

See also:

■ Detach Raster Images on page 912
Load and Unload Raster Images

You can improve performance by unloading images when you do not need them in the current drawing session. Unloaded images are not displayed or plotted; only the image boundary is displayed. Unloading an image does not alter its link. If memory is not sufficient to open multiple attached images in a drawing, images are automatically unloaded.

In the Reference Manager, you can use Reload to reload an unloaded image or to update a loaded image by reloading the image from the specified directory path. If a drawing is closed after an image is unloaded, the image file is not loaded when the drawing is next opened; you must reload it.

Quick Reference

Commands
EXTERNALREFERENCES
Opens the Reference Manager palette.
PROPERTIES
Controls properties of existing objects.

Improve the Display Speed of Raster Images

To increase the display speed of images, you can change image display quality, hide images not currently needed, use image tiling, or suppress image selection highlighting.

To increase the display speed of images, you can change image display quality from the default high quality to draft quality. Draft-quality images appear more grainy (depending on the image file type), but they are displayed more quickly than high-quality images. Use the IMAGEQUALITY system variable to control image quality.

You can improve the image quality when using True Color (24 or 32 bits per pixel) for raster images by setting certain drafting environment options. When images are displayed at optimum quality, regeneration time increases significantly. To improve performance, decrease the number of colors for the system display setting while working in a drawing.

You can increase redrawing speed by hiding images you do not need in the current drawing session. Hidden images are not displayed or plotted; only the
drawing boundary is displayed. You can choose to hide an image regardless of the user coordinate system (UCS) in the current viewport.

**Use Tiled Images**

Tiled images are small portions (a series of tiles) of large images that load much faster than non-tiled images. If you edit or change any properties of an image, only the modified portion is regenerated, thus improving the regeneration time. TIFF (Tagged Image File Format) is the only tiled format that the program supports. The TIFF reader supports all image types:

- Bitonal (1 bit per pixel)
- Gray scale and indexed color (8 bits per pixel)
- True Color (24 or 32 bits per pixel)

You can save tiled TIFF images with most image scanning tools. The image tiles should be no smaller than 64 x 64 pixels and no larger than 512 x 512 pixels. Additional file readers that support other tiled formats, such as CALS Type II, are available from third-party developers.

**Suppress Highlighting When Selecting Images**

You can turn on or off the highlighting that identifies the selection of a raster image or the image frame by toggling the value of IMAGEHLT system variable. By default, IMAGEHLT is set to 0, to highlight only the raster image frame. Turning off highlighting of the entire image improves performance.

**Quick Reference**

**Commands**

- **IMAGEQUALITY**
  Controls the display quality of images.

- **PROPERTIES**
  Controls properties of existing objects.

**System Variables**

- **IMAGEHLT**
  Controls whether the entire raster image or only the raster image frame is highlighted.
Export Drawings to Other File Formats

If you need to use the information from a drawing file in another application, you can convert it to a specific format by exporting it.

Export PDF Files

You can export a drawing as a PDF file to facilitate sharing information with other design groups.

Sets of drawings are the primary deliverable for most design groups. Creating a drawing set to distribute for review can be complicated and time consuming. Electronic drawing sets are saved as PDF files.

To output a single drawing as a PDF, use the Print dialog box.

Quick Reference

Commands

EXPORT
Saves the objects in a drawing to a different file format.

PLOT
Outputs a drawing to a printer or file.

Export DXF Files

You can export a drawing as a DXF file, which contains drawing information that can be read by other CAD systems.

You can export a drawing as a DXF (drawing interchange format) file. DXF files are text or binary files that contain drawing information that can be read by other CAD programs. If you are working with consultants who use a CAD program that accepts DXF files, you can share a drawing by saving it as a DXF file.

You can control floating-point precision of the DXF format up to 16 decimal places and save the drawing in either ASCII or binary format. ASCII format results in a text file that you can read and edit; binary format results in a significantly smaller file that is faster to work with.
If you do not want to save the entire drawing, you can choose to export selected objects only. You can use this option to remove extraneous material from drawing files.

Quick Reference

Commands

EXPORT
- Saves the objects in a drawing to a different file format.

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

Export Raster Files

You can create a device-independent raster image of the objects in your drawing.

Several commands can be used to export objects into device-independent raster images in the bitmap, JPEG, TIF, and PNG formats.

Objects are displayed in the raster image as they appear on the screen, including objects in shaded and rendered viewports.

Objects are displayed in the raster image as they appear on the screen, including objects in shaded viewports.

File formats such as JPEG are compressed as they are created. Compressed files take up less disk space, but they might not be readable by certain applications.

Quick Reference

Commands

BMPOUT
- Saves selected objects to a file in device-independent bitmap format.

EXPORT
- Saves the objects in a drawing to a different file format.

JPGOUT
- Saves selected objects to a file in JPEG file format.
PNGOUT
Saves selected objects to a file in a Portable Network Graphics format.

TIFOUT
Saves selected objects to a file in TIFF file format.

Export PostScript Files

You can convert a drawing file to a PostScript file, a format that is used by many desktop publishing applications.

The PostScript file format type is used by many desktop publishing applications. Its high-resolution print capabilities make it preferable to raster formats, such as GIF, PCX, and TIFF. By converting the drawing to a PostScript format, you can also use PostScript fonts.

Export in PostScript Format

When you export a file in PostScript format as an EPS file, some objects are handled specially.

- **Thickened text, text control codes.** If text has a thickness greater than 0 or contains control codes (such as "%%O" or "%%D"), it is not plotted as PostScript text, although the text is accurately plotted. International and special symbols (such as "%%213") are output as PostScript text.

- **ISO 8859 Latin/1 character set.** When text uses character codes in the 127 to 255 range, the text is interpreted according to the ISO 8859 Latin/1 character set. If such a character appears in text that is mapped to PostScript, a version of the font is generated with an encoding vector remapped to represent the ISO character set. The resulting text is output in PostScript in a form compatible with the font.

- **Circles, arcs, ellipses, elliptical arcs.** Except when they have thickness, arcs and circles are translated into the equivalent PostScript path objects.

- **Filled solids.** A solid fill is plotted as a PostScript filled path.

- **Two-dimensional polylines.** A 2D (planar) polyline with uniform width is output as a PostScript stroked path. The PostScript end cap and miter limit variables are set to approximate the segment joining.
Quick Reference

Commands

EXPORT
Saves the objects in a drawing to a different file format.

PLOT
Outputs a drawing to a printer or file.

Export ACIS SAT Files

You can export certain object types to an ACIS file in ASCII (SAT) format.
You can export trimmed NURBS surfaces, regions, and 3D solids to an ACIS file in ASCII (SAT) format. Other objects, such as lines and arcs, are ignored.
Use the ACISOUTVER system variable to specify the ACIS version for the output of the ACISOUT command.

Quick Reference

Commands

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

EXPORT
Saves the objects in a drawing to a different file format.

System Variables

ACISOUTVER
Controls the ACIS version of SAT files created using the ACISOUT command.

Export Stereolithography STL Files

You can export 3D solid objects in the STL file format compatible with stereolithography or 3D printing.
Stereolithography or 3D printing is used in rapid prototyping, rapid manufacturing, and other applications to create physical parts and models.
The 3D solid data is translated to a faceted mesh representation consisting of a set of triangles and saved to an STL file. Use the FACETRES system variable to adjust the facet density to an appropriate detail level.

**NOTE** Setting the resolution too high slows down the fabrication process without improving the output quality of the stereolithography device.

The STL data is used to create an object by depositing a succession of thin layers of plastics, metals, or composite materials. The resulting parts and models are commonly used for the following:

- Visualize design concepts
- Create product mockups, architectural models, and terrain models
- Test form, fit, and function
- Identify design problems
- Create masters for vacuum forming applications
- Create Marketing tools

**Quick Reference**

**Commands**

**EXPORT**

Saves the objects in a drawing to a different file format.

**System Variables**

**FACETRES**

Adjusts the smoothness of shaded and rendered objects and objects with hidden lines removed.

**Use Drawings from Different Versions and Applications**

You can share drawing files from AutoCAD and AutoCAD LT, drawing files from previous versions, and drawing files that contain custom objects. In some cases there are limitations.
Work with Drawings in Earlier Releases

When you work with drawings created in AutoCAD 2008 (and later releases) in AutoCAD 2007 (and earlier releases), you should be aware of the following visual fidelity issues.

Visual Fidelity for Annotative Objects in Previous Releases

You can specify that objects maintain visual fidelity when they are viewed in AutoCAD 2007 (and earlier releases) with the SAVEFIDELITY system variable. If you work primarily in model space, it is recommended that you turn off visual fidelity (set SAVEFIDELITY to 0). However, if you need to exchange drawings with other users, and layout fidelity is most important, then visual fidelity should be turned on (SAVEFIDELITY set to 1).

Annotative objects may have multiple . When visual fidelity is on, annotative objects are decomposed and scale representations are saved (in an ) to separate layers. These layers are named based on their original layer and appended with a number. If you explode the block in AutoCAD 2007 (or earlier releases), and then open the drawing in AutoCAD 2008 (or later releases), each scale representation becomes a separate annotative object, each with one annotation scale. It is recommended that you do not edit or create objects on these layers when working with a drawing created in AutoCAD 2008 (and later releases) in AutoCAD 2007 (and earlier releases).

When visual fidelity for annotative objects is not selected, a single model space representation is displayed on the Model tab. Depending on the setting of the ANNOALLVISIBLE system variable, more annotation objects may be displayed on the Model tab, and more objects may be displayed in paper space viewports at different sizes than in AutoCAD 2008 and later releases.

Annotative Object Properties in Previous Releases

In an AutoCAD 2008 drawing, when an annotative block does not have its paper orientation set to match the layout, and the block contains multiline attributes that are based on a text style that is not set to match the orientation of the layout, the attributes may shift positions if you open this drawing in AutoCAD 2007 (and earlier releases).

Layer Property Overrides in Previous Releases

When you open an AutoCAD 2008 drawing containing layer property overrides, overrides are not visible. The property override settings are retained when the drawing is saved in a previous release, and are visible again when the drawing is opened in AutoCAD 2008.
If a viewport containing layer property overrides is deleted when the drawing is opened in a previous release, the override settings are not retained and are not available when the drawing is opened in AutoCAD 2008.

When the VISRETAIN system variable is set to 0 when the drawing is opened in a previous release, xref layers containing viewport property overrides are not retained.

If you open an AutoCAD 2008 drawing in a previous release, property overrides may display in a thumbnail image. When the drawing is saved with a layout tab, and then opened in the previous release, those property overrides do not display.

**Dimension Enhancements in Previous Releases**

AutoCAD 2008 dimension enhancements are lost when they are edited in earlier releases. If you don’t change these dimensions, they are restored when you open the drawing in AutoCAD 2008.

The following dimension enhancements do not lose visual fidelity in previous releases if they are not edited:

- Dimension breaks
- Jogged linear dimensions
- Inspection dimensions
- Angular dimensions that are dimensioned using the quadrant option
- Arc extension lines for radial and diameter dimensions

**Multileader Objects in Previous Releases**

Multileaders display as proxy objects in releases prior to AutoCAD 2008. The PROXYSHOW system variable controls the display of proxy objects in a drawing.

**MTEXT Paragraph and Paragraph Line Spacing in Previous Releases**

Some of the new paragraph spacing and paragraph line spacing options are not supported when an AutoCAD 2008 mtext object is opened in AutoCAD 2007 (and earlier releases).

The following mtext formatting features have no visual fidelity in previous releases:

- Paragraphs with justified alignment
Paragraphs with distributed alignment
Fields that wrap across columns
Fields that wrap across lines that have new paragraph alignments
Paragraphs with non-default alignments in mtext without left object-level justification

The following mtext formatting features have some visual fidelity in previous releases (when it’s possible to add white spaces or replace text with white spaces):

Paragraphs with non-default alignments (other than justified or distributed) in mtext that has left object-level justification
Paragraphs with tabs using new tab alignments (center, right, or decimal alignment applied)
Paragraphs with new line spacing that can be "approximated" with "tall" spaces

Mtext with new formatting that is edited and saved in previous releases loses the new formatting when re-opened in AutoCAD 2008.

Tables in Previous Releases
Editing AutoCAD 2008 tables in previous releases removes AutoCAD 2008 table formatting. Also, AutoCAD 2008 table cells with long block and text strings may extend outside of cell borders when opened in previous releases.

Multiple-Language Support in Previous Releases
Drawing properties in AutoCAD 2008 are saved with Unicode characters. For instance, if you save the latest format drawing containing multiple language drawing properties to a 2004-format drawing, the drawing properties are converted to the native characters of the current Windows language. If text cannot be converted to the native characters, it is saved to CIF codes (\U+nnnn) or MIF codes (\M+nxxxx).

When saving the latest format drawing to a 2004-format drawing, any new symbol or dictionary names (for example, layout name, text style name, dimension style name) created in AutoCAD 2008 are saved in the language that was used when the symbol names were created.

In order to view and edit drawings with characters that are not included in the languages specified for your operating system, make sure supplemental
language support is installed in your computer operating system. You can specify the language in the Regional and Language Options dialog box, available from the Windows Control Panel. (You may be able to view text that uses SHX fonts without specifying extra language support.)

Text styles for Asian languages that use SHX and Big Font can support characters only from the same code page. For example, text styles that use a Japanese Big Font cannot support German or Korean characters. (English characters, which are part of every code page, are supported.) Multiple-language support for non-Asian languages is supported for text styles that use SHX fonts with Big Fonts disabled. (The SHX font must define the required characters.)

Multiple-language support does not exist in some earlier versions of AutoCAD. For example, when you save a file to AutoCAD 2000 format, the contents of multiple-language multiline text may be corrupted. This problem is more likely to happen when you open and save a drawing on an operating system with a system language setting that differs from the system in which the drawing was last saved.

**NOTE** Drawings that include external references (xrefs) to drawing files saved in earlier releases also have the limitations described above.

---

**Save Drawings to Previous Drawing File Formats**

You can save a drawing in a format compatible with previous versions of the product.

You can save a drawing created with the current release of the program in a format compatible with previous versions. This process creates a drawing with information specific to the current release stripped out or converted to another object type.

If you use the current release to open a drawing created with a previous release, and you do not add any information specific to the current release, you can then save the drawing in the format of the previous release without loss of data.

**NOTE** To use files with AutoCAD Release 12 or AutoCAD LT Release 2, save the drawing using the AutoCAD R12/LT2 DXF option.

If you need to keep a drawing created in a previous release in its original format, either mark the file as read-only, or open it in the current release and use the File Type options in the Save As dialog box to save it in its original format.
Because saving a drawing in an earlier release format may cause some data loss, be sure to assign a different name to avoid overwriting the current drawing. If you overwrite the current drawing, you can restore the overwritten version from the backup file (filename.bak) that is created during the saving process.

Maintain Associativity in Dimensions

Associative dimensions created in AutoCAD 2002 or later generally maintain their associativity when saved to a previous release and then reopened in the current release. However, if you modify dimensioned objects using a previous release to the extent that new objects are formed, the dimension associations change when the drawing is loaded into the current release. For example, if a line that was dimensioned is trimmed so that an interior portion of the line is removed, two line objects result and the associated dimension applies to only one of the line objects.

Dimension associativity is not maintained when a drawing is saved as an AutoCAD R12/LT 2 DXF file and then reopened in the current release.

Save Drawings with Large Objects

Drawings saved to a legacy drawing file format (AutoCAD 2007 or earlier) do not support objects greater than 256MB. For more information about saving drawings that contain large objects to a previous release, see Maintain Compatibility with Large Object Limits on page 70.

Limitations of Saving to Earlier Versions

Saving a drawing in Release 2000/LT 2000 format is subject to the following limitations:

■ File size can increase.

■ Encryption and digital signatures are not preserved.

Saving a drawing in Release 14/LT 98/LT 97 format is subject to the following limitations:

■ Hyperlinks are converted to Release 14/LT 98/LT 97 attached URLs.

■ Database links and freestanding labels are converted to Release 14/LT 98/LT 97 links and displayable attributes.
- Database attached labels are converted to multiline text and leader objects, and their link information is not available. Attached labels are restored if you open the drawing in AutoCAD 2000 or later.

- Dynamic block geometry can be redefined independent of the block's dynamic elements, and the geometry in the block reference is not updated when the drawing is opened in AutoCAD 2011 for Mac or later.

- Dimensions created using the DIMARC and DIMJOGGED commands may not retain their original color in Release 14/LT 98/LT 97.

Saving a drawing in Release 12/LT 2 DXF format is subject to the following limitations:

- Lightweight polylines and hatch patterns are converted to Release 12 polylines and hatch patterns.

- All solids, bodies, regions, ellipses, leaders, multilines, rays, tolerances, and xlines are converted to lines, arcs, and circles as appropriate.

- Groups, complex linetypes, OLE objects, and preview images are not displayed.

- Many objects are lost if you save a drawing as Release 12 and open it in Release 2000/LT 2000 or later.

- Multiple layouts and layout names are lost. Only the Model tab and the current layout tab are saved.

- Spaces in the names of layers and other objects are converted to underscores, and their maximum length is 32 characters.

- DWF or DWFx underlay files attached to drawings cannot be saved to Release 12/LT 2 DXF format.

- The status of external references as unloaded is lost.

**Quick Reference**

**Commands**

**CONVERT**

Optimizes 2D polylines and associative hatches created in AutoCAD Release 13 or earlier.
OPTIONS
Customizes the program settings.

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

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

System Variables

PLINETYPE
Specifies whether optimized 2D polylines are used.

Work with AutoCAD Drawings in AutoCAD LT
AutoCAD LT offers full compatibility when working with AutoCAD drawings. However, you should understand how AutoCAD LT handles AutoCAD-only features.

NOTE This topic compares the full feature set of AutoCAD and AutoCAD LT in the Windows environment. Currently, AutoCAD for the Mac also has feature set limitations, but the compatibility of drawing files between all products is designed to avoid loss of any information.

Work with Fields
In AutoCAD, you can create a sheet set, and insert LispVariables and SheetSet Manager fields. The LispVariables and SheetSet Manager fields are not available in AutoCAD LT. The drawings created in AutoCAD that contain LispVariables or SheetSet Manager fields can be opened without errors in AutoCAD LT and the cached value is displayed.

Work with Multiple User Coordinate Systems
In AutoCAD, you can choose to use a different user coordinate system (UCS) in each viewport in a single drawing file. In AutoCAD LT, you can use only one UCS in each drawing file. The AutoCAD LT behavior is the same as it was in previous releases.

When you open an AutoCAD drawing file in AutoCAD LT, AutoCAD LT uses only the UCS from the current viewport. If you edit the drawing in AutoCAD LT, and then save it and reopen it in AutoCAD, you may notice some
discrepancies in UCS usage. User coordinate systems that were set individually in AutoCAD will probably change if the viewports that use them were activated in the AutoCAD LT session.

**Work with AutoCAD for Mac 2D and 3D Solid Object Shading**

In AutoCAD, visual styles provide shading and wireframe options for objects in the current viewport. AutoCAD LT does not support visual styles. The SHADEMODE command in AutoCAD LT provides only the 2D Wireframe and Hidden options. You can use SHADEMODE in AutoCAD LT to turn off visual styles in viewports that were created in AutoCAD. This exposes the underlying geometry so you can easily edit drawings and use the geometry with precision drawing tools such as object snaps.

**WARNING** Once you use the SHADEMODE command in AutoCAD LT to turn solid object shading off for an object created in AutoCAD, you cannot turn it back on except by using the UNDO command. If you make changes to the object, you can turn the shading on again only in AutoCAD.

**Work with Constraints**

Some of the drawings that you work with will contain design requirements enforced within the drawing itself through the use of constraints. Using constraints, you can enforce requirements while experimenting with different designs.

A constrained object will move in a predictable manner when edited or moved. A single variable change can cause all related objects to change automatically, enabling you to run through design iterations simply and effectively.
There are two general types of constraints supported: Geometric and Dimensional.

- Geometric constraints determine the relationships between 2D geometric objects or points on objects relative to each other. Use constraint bars to view the geometric constraints applied to objects. Constraint bars are visible only when you place your cursor over the highlighted nodes.

- Dimensional constraints control distances or angles between 2D geometric objects in a drawing. The main dimensional constraints are: dynamic, annotational, and reference constraints.
  - Dynamic constraints (default) - Used to constrain objects and are displayed on demand.
  - Annotational constraints - Used to create associative variables, offset distances, and so on.
  - Reference constraints (read-only) - Read-only dimensional constraints (either dynamic or annotational).

When you place your cursor over a constrained object, you will see a glyph denoting the object is constrained.

With AutoCAD LT, you can do the following:

- View drawings containing constraints created using AutoCAD.
- View and edit the geometric and dimensional constraints.

**NOTE** You cannot create constraints within AutoCAD LT.

**Work with Dynamic Blocks**

In AutoCAD, you can add new constraint parameters to a dynamic block in the Block Editor. In AutoCAD LT, you can open drawings which include a block definition containing geometric or dimensional constraints or a block properties table but cannot add new parameters to the block definition.

When you work with constrained dynamic blocks in the Block Editor in AutoCAD LT, you can modify the constraint values and delete constraints but cannot add constraints in the Block Editor. You can manage the constraints while in the Block Editor from the Parametric tab.
Modify 3D Point Clouds

While you cannot index or attach a point cloud in AutoCAD LT, you can open drawings created in AutoCAD that contain point clouds. When an attached point cloud file is unlocked, you can select the point cloud with a grip that is displayed at the centroid of the point cloud.

NOTE You cannot explode a point cloud.

Turn Off Perspective View in an AutoCAD Drawing

Set the PERSPECTIVE system variable to 0 to turn off perspective view in an AutoCAD drawing that is open in AutoCAD LT. You cannot turn on perspective view in a drawing that is open in AutoCAD LT.

Work with Custom and Proxy Objects

Custom objects provide additional capabilities to the program and related products. When the application that created the custom object is not available, a proxy object is substituted in its place.

A custom object is a type of object created by an ObjectARX® (AutoCAD Run-Time Extension) application, which typically has more specialized capabilities than standard AutoCAD for Mac objects. Custom objects include parametric solids (AutoCAD® Mechanical), intelligently interactive door symbols (AutoCAD® Architecture), polygon objects (AutoCAD® Map 3D), and associative dimension objects (AutoCAD and AutoCAD LT).

In addition to Autodesk, many software vendors use ObjectARX to write programs that create graphical and nongraphical custom objects that are useful in their AutoCAD based applications.

Proxy Objects

A proxy object is a substitute for a custom object when the ObjectARX application that created the custom object is not available to AutoCAD for Mac or other host applications. Later, when the application is available, the proxy object is replaced by the custom object.

Proxy objects have significantly reduced capabilities compared to their corresponding custom objects. The extent to which proxy objects can be edited is determined by the parent ObjectARX application. For example, operations such as erasing and moving an object, or changing object properties, may or may not be possible on a proxy object, depending on the application that created it.
When you open a drawing, you might see a Proxy Information dialog box. The dialog box tells you the total number of proxy objects in the drawing (both graphical and nongraphical) and the name of the missing application and provides additional information about the proxy object type and display state. You can use the dialog box to control the display of proxy objects.

**Object Enablers**

An object enabler is a tool that provides specific viewing and standard editing access to a custom object in the host applications when the application that created the custom object is not present.

Object Enablers allow custom objects in a drawing to behave with more intelligence than proxy graphics. Object enablers also facilitate workgroup collaboration when using other Autodesk products.

If the ObjectARX application is not installed on your system, you can check for available Object Enablers on the Web. For example, if you receive a drawing that contains objects that were created in AutoCAD Architecture, but you don’t have that application installed on your system, the AEC Object Enabler is downloaded so you can view those drawings as they were intended.

For a complete list of the currently available Object Enablers, go to the Autodesk Web site at [http://www.autodesk.com/enablers](http://www.autodesk.com/enablers).

**Object Classification**

If an application such as AutoCAD Map 3D was used to add a feature (object) classification to an object, you can view the classification in the Properties Inspector palette in the Class Name entry. If the object’s classification is missing from the associated classification (XML) file, or if the associated classification file is missing, an exclamation mark is displayed in the Class Name entry. For information about what is required to create a feature classification, see the AutoCAD Map 3D documentation.

**Quick Reference**

**Commands**

EXPORTTOAUTOCAD

Creates a new DWG file with all AEC objects exploded.
**System Variables**

PROXYGRAPHICS

Specifies whether images of proxy objects are saved in the drawing.

PROXYNOTICE

Displays a notice when a proxy is created.

PROXYSHOW

Controls the display of proxy objects in a drawing.
Collaborate with Others
Use the Internet for Collaboration

You can access and store drawings and related files on the Internet.

Get Started with Internet Access

Before you can transfer or save files to an Internet or an intranet location, you have to get access permissions and take security precautions.

In this topic and others, the term Internet is used to refer to both the Internet and an intranet. An intranet is a private network that uses the same standards as the Internet.

To save files to an Internet location, you must have sufficient access rights to the directory where the files are stored. Contact your network administrator or Internet service provider (ISP) to receive access rights for you and anyone else who needs to work with the files.

If you connect to the Internet through your company's network, you might have to set up a proxy server configuration. Proxy servers act as security barriers by shielding information on your company's network from potential security risks due to external Internet access.

Contact your network administrator for details about how to configure a proxy server in your network environment.
Quick Reference

Commands

BROWSER
Launches the default web browser defined in your system's registry.

System Variables

INETLOCATION
Stores the Internet location used by the BROWSER command and the Browse the Web dialog box.

Work with Drawing Files over the Internet

You can open and save drawings to an Internet location, attach externally referenced drawings stored on the Internet, use i-drop to insert blocks by dragging drawings from a website, and create a transmittal package of drawings that automatically includes all related files.

Open and Save Drawing Files from the Internet

The file input and output commands recognize any valid Uniform Resource Locator (URL) path to a DWG file.

You can use AutoCAD for Mac to open and save files from the Internet. The AutoCAD for Mac file input and output commands (OPEN, EXPORT, and so on) recognize any valid URL path to an AutoCAD for Mac file. The drawing file that you specify is downloaded to your computer and opened in the AutoCAD for Mac drawing area. You can then edit the drawing and save it, either locally or back to any Internet or intranet location for which you have sufficient access privileges.

If you know the URL to the file you want to open, you can enter it directly in the Select File dialog box. You can also browse defined FTP sites or web folders in the Select File dialog box.
Quick Reference

Commands

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

EXPORT
Saves the objects in a drawing to a different file format.

OPEN
Opens an existing existing drawing file.

Share Drawing Files Internationally

Beginning with AutoCAD 2007-based products, drawing files and most files associated with drawing files use the Unicode standard. This lets you maintain both the visual fidelity and data integrity of international characters when you save and open drawing files.

NOTE AutoCAD 2006, AutoCAD LT 2006, and prior versions were not Unicode applications. When sharing drawings with earlier, non-Unicode, versions, use ASCII characters to ensure compatibility when you save files, insert xrefs, and specify folder paths.

Overview of Unicode

All characters are processed numerically by the computer operating system, which assigns a number to each character. Various numeric encoding systems have been used in the past, however these encoding systems often conflicted. As a result, operating systems and applications relied on code pages with specific character sets and numbering assigned to countries or regions.

To facilitate international compatibility, the Unicode standard was adopted by major industry leaders and is being maintained by the Unicode Consortium.

Drawing File Impact

Language-specific characters can be used in file names and text within drawing files, or files associated with drawing files. The following are common examples:

- Drawing file names
■ Folder path names
■ Named objects such as layers and blocks within a drawing
■ Linetype and hatch pattern file names and their contents
■ Text used in notes and dimensions within a drawing

This means that drawings can be opened, worked on, and saved worldwide regardless of language-specific characters. The only requirement is that the appropriate language pack must be installed first.

When you save text files such as linetype (LIN), hatch pattern (PAT), and script (SCR) files using an application such as Notepad, it is recommended that you specify Unicode encoding at the bottom of the Save As dialog box to ensure compatibility.

Limitations

Most international drawing projects can be completed within the current product environment. However, there are several file types and features that are not supported yet between countries and regions that use different Windows code pages. These features include the following:

■ Round trip file and data compatibility with non-Unicode products
■ Block attributes
■ Visual LISP files

Quick Reference

Commands

OPEN
  Opens an existing drawing file.

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

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

**TEXTOUTPUTFILEFORMAT**

Provides Unicode options for log files.

---

**Access Buzzsaw for Project Collaboration**

Using Autodesk® Buzzsaw® you can store, manage, and share documents that populate a Buzzsaw site.

Buzzsaw is a secure, online project collaboration service in which members in different locations can post files to and access files from a centralized site. You can save files, send transmittal sets, and publish sheets to Buzzsaw.

You can get a free 30-day trial subscription when you follow the procedures to access Buzzsaw, or from the Autodesk website. Your Buzzsaw subscription includes user licenses to access Buzzsaw. When you invite your consultants and contractors to your site, the Buzzsaw software is automatically downloaded.

**Prepare to Use Buzzsaw**

To use Buzzsaw, you must already have a project hosting account or be given access to a subscriber's Buzzsaw site. The subscriber will provide you with the Buzzsaw URL, user name, and password so that you can log in, access, and post files.

**Use Buzzsaw**

You can access Buzzsaw from several standard file selection dialog boxes (such as New or Open). The Buzzsaw icon is displayed in the Places list for quick access.

Support for Autodesk Buzzsaw services includes the following features:

If you plan to use Buzzsaw for project collaboration, set up some shortcuts so you can navigate to frequently used sites more quickly. Using the Buzzsaw icon in the Places list, you can specify a shortcut to an existing project collaboration site from the list or create a new shortcut using the Add a Buzzsaw Location Shortcut.

You can right-click in the Location area of a dialog box that supports Buzzsaw to display a shortcut menu that provides options to add, manage, or delete Autodesk Buzzsaw locations. Changes you make to locations get stored in the registry.
Quick Reference

Commands
OPEN
Opens an existing drawing file.

Work with Xrefs over the Internet
You can attach externally referenced drawings stored on the Internet or an intranet to drawings stored locally on your system.

For example, you might have a set of construction drawings that are modified daily by a number of contractors. These drawings are stored in a project directory on the Internet. You can maintain a master drawing on your computer, and attach the Internet drawings to the master drawing as external references (xrefs). When any of the Internet drawings are modified, the changes are included in your master drawing the next time you open it. This is a powerful mechanism for developing accurate, up-to-date composite drawings that can be shared by a design team.

NOTE If you have a slow Internet connection or are working with a master drawing that has many xrefs attached, the download of the xrefs to your system might take a long time.
Render Drawings
Draw 2D Isometric Views

The Isometric Snap/Grid mode helps you create 2D isometric images that represent 3D objects. By setting the Isometric Snap/Grid, you can easily align objects along one of three isometric planes; however, although the isometric drawing appears to be 3D, it is actually a 2D representation. Therefore, you cannot expect to extract 3D distances and areas, display objects from different viewpoints, or remove hidden lines automatically.

Set Isometric Grid and Snap

Simulate a 3D object from a particular viewpoint by aligning along three major axes.

Isometric drawings simulate a 3D object from a particular viewpoint by aligning along three major axes.

By setting the Isometric Snap/Grid, you can easily align objects along one of three isometric planes; however, although the isometric drawing appears to be 3D, it is actually a 2D representation. Therefore, you cannot expect to extract 3D distances and areas, display objects from different viewpoints, or remove hidden lines automatically.

If the snap angle is 0, the axes of the isometric planes are 30 degrees, 90 degrees, and 150 degrees. Once you set the snap style to Isometric, you can work on any of three planes, each with an associated pair of axes:

- **Left.** Aligns snap and grid along 90- and 150-degree axes.
- **Top.** Aligns snap and grid along 30- and 150-degree axes.
- **Right.** Aligns snap and grid along 30- and 90-degree axes.
Choosing one of the three isometric planes causes Ortho and the crosshairs to be aligned along the corresponding isometric axes. For example, when Ortho is on, the points you specify align along the simulated plane you are drawing on. Therefore, you can draw the top plane, switch to the left plane to draw another side, and switch to the right plane to complete the drawing.

**Quick Reference**

**Commands**

**DSETTINGS**

Sets grid and snap, polar and object snap tracking, object snap modes, and Dynamic Input.

**GRID**

Displays a grid pattern in the current viewport.

**ISOPLANE**

Specifies the current isometric plane.
ORTHO
Constrains cursor movement to the horizontal or vertical direction.

SNAP
Restricts cursor movement to specified intervals.

**System Variables**

SNAPISOPAIR
Controls the isometric plane for the current viewport.

SNAPSTYL
Sets the snap style for the current viewport.

---

**Draw Isometric Circles**

Represent circles on isometric planes using ellipses.

If you are drawing on isometric planes, use an ellipse to represent a circle viewed from an oblique angle. The easiest way to draw an ellipse with the correct shape is to use the Isocircle option of ELLIPSE. The Isocircle option is available only when the Style option of Snap mode is set to Isometric (see DSETTINGS).

**NOTE** To represent concentric circles, draw another ellipse with the same center rather than offsetting the original ellipse. Offsetting produces an oval-shaped spline that does not represent foreshortened distances as you would expect.

---

**Quick Reference**

**Commands**

ELLIPSE
Creates an ellipse or an elliptical arc.

ISOPLANE
Specifies the current isometric plane.
Add Lighting to Your Model

Lighting can be added to a scene to create a more realistic rendering.

Overview of Lighting

Lighting adds the finishing touch to the scene.

Default Lighting

When there are no lights in a scene, the scene is shaded with default lighting. Default lighting is derived from two distant sources that follow the viewpoint as you move around the model. All faces in the model are illuminated so that they are visually discernible. You can control brightness and contrast, but you do not need to create or place lights yourself.

When you insert custom lights or add sunlight, you can disable the default lighting. You can apply default lighting to the viewport only; at the same time, you can apply custom lights to the rendering.

Standard Lighting Workflow

You add lights to give the scene a realistic appearance. Lighting enhances the clarity and three-dimensionality of a scene. You can create point lights, spotlights, and distant lights to achieve the effects you want. You can move or rotate them with grip tools, turn them on and off, and change properties such as color and attenuation. The effects of changes are visible in the viewport in real time.

Spotlights and point lights are each represented by a different light glyph (a symbol in the drawing showing the location of the light). Distant lights and the sun are not represented by glyphs in the drawing because they do not have
a discrete position and affect the entire scene. You can turn the display of light glyphs on or off while you work. By default, light glyphs are not plotted.

**Photometric Lighting Workflow**

For more precise control over lighting, you can use photometric lights to illuminate your model. Photometric lights use photometric (light energy) values that enable you to define lights more accurately as they would be in the real world. You can create lights with various distribution and color characteristics, or import specific photometric files available from lighting manufacturers.

Photometric lights can use manufacturers’ IES standard file format. By using manufacturers’ lighting data, you can visualize commercially available lighting in your model. Then you can experiment with different fixtures, and by varying the light intensity and color temperature, you can design a lighting system that produces the results you want.

**Luminaire Objects**

Light fixtures can be represented by embedding photometric lights in blocks that also contain geometry. A luminary assembles a set of light objects into a light fixture.
Quick Reference

Commands

DISTANTLIGHT
  Creates a distant light.

LIGHT
  Creates a light.

FREESPOT
  Creates free spotlight which is similar to a spotlight without a specified target.

FREEWEB
  Creates a free web light which is similar to a web light without a specified target.

POINTLIGHT
  Creates a point light that radiates light in all directions from its location.

SPOTLIGHT
  Creates a spotlight that emits a directional cone of light.

TARGETPOINT
  Creates a target point light.
WEBLIGHT

 Creates a web light.

**System Variables**

DEFAULTLIGHTING

 Turns on and off default lighting in place of other lighting.

DEFAULTLIGHTINGTYPE

 Specifies the type of default lighting, old or new.

LIGHTGLYPHDISPLAY

 Turns on and off the display of light glyphs.

LIGHTINGUNITS

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

LIGHTSINBLOCKS

 Controls whether lights contained in blocks are used when rendering.

LINEARBRIGHTNESS

 Controls the brightness level of the viewport when using default lighting or generic lights.

LINEARCONTRAST

 Controls the contrast level of the viewport when using default lighting or generic lights.

RENDERUSERLIGHTS

 Controls whether to override the setting for viewport lighting during rendering.

SUNSTATUS

 Turns on and off the lighting effects of the sun in the current viewport.

**Standard and Photometric Lighting Workflow**

 Types of lighting selected globally affect a drawing.
Set the Type of Lighting

AutoCAD for Mac offers three choices for lighting units: standard (generic), International (SI), and American. The standard (generic) lighting workflow is equivalent to the lighting workflow in AutoCAD for Mac prior to AutoCAD 2008. The default lighting workflow for drawings created in AutoCAD 2008 and later is a photometric workflow based on International (SI) lighting units. This choice results in physically correct lighting. The American lighting unit provides another option. American differs from International in that illuminance values are formatted in foot-candles rather than lux.

In previous versions of AutoCAD for Mac, standard lighting was the default. You can change the type of lighting with the LIGHTINGUNITS system variable. The LIGHTINGUNITS system variable set to 0 represents standard (generic) lighting; set to 1 represents photometric lighting in American units; set to 2 represents photometric lighting in International SI units.

Photometric Lights

Photometric lights are physically correct lights. Physically correct lights attenuate as the square of the distance.

You can create lights with various distribution and color characteristics, or import specific photometric files available from lighting manufacturers. Photometric lights always attenuate using an inverse-square falloff, and rely on your scene to use realistic units.

Tone Mapping

Generally with photometric lights and especially the sun you need to perform tone mapping. Tone mapping can be adjusted with the RENDEREXPOSURE command. The Adjust Rendered Exposure dialog box provides a preview and controls to adjust the tone mapping.

Quick Reference

System Variables

LINEARBRIGHTNESS

Controls the brightness level of the viewport when using default lighting or generic lights.

LINEARCONTRAST

Controls the contrast level of the viewport when using default lighting or generic lights.
Illuminate a Scene

You can add point lights, spotlights, and distant lights and set the location and photometric properties of each.

You can use a command to create a light, or you can use a button on the Lights toolbar or the Lights panel on the ribbon. You can use the Properties Inspector palette to change the color of a selected light or other properties. You can also store a light and its properties on a tool palette and use it again in the same drawing or another drawing.

Guidelines for Lighting

The guidelines for lighting used by photographers, filmmakers, and stage designers can help you set up the lighting for scenes.

Your choice of lighting depends on whether your scene simulates natural or artificial illumination. Naturally lit scenes, such as daylight or moonlight, get their most important illumination from a single light source. Artificially lit scenes, on the other hand, often have multiple light sources of similar intensity.

Natural Light

For practical purposes at ground level, sunlight has parallel rays coming from a single direction. The direction and angle vary depending on the time of day, the latitude, and the season.

In clear weather, the color of sunlight is a pale yellow: for example, RGB values of 250, 255, 175 (HSV 45, 80, 255). Cloudy weather can tint sunlight blue, shading into dark gray for stormy weather. Particles in the air can give sunlight an orange or brownish tint. At sunrise and sunset, the color can be more orange or red than yellow.

Shadows are more distinct the clearer the day is, and can be essential for bringing out the three-dimensionality of a naturally lit scene.

A directional light can also simulate moonlight, which is white but dim compared to the sun.

Artificial Light

A scene illuminated by point lights, spotlights, or distant lights is artificially illuminated. Therefore, it can be helpful to know how light behaves.
When light rays strike a surface, the surface reflects them, or at least some of them, enabling us to see the surface. The appearance of a surface depends on the light that strikes it combined with the properties of the surface material, such as color, smoothness, and opacity.

Other factors, such as a light’s color, intensity, attenuation, and angle of incidence also play a role in how objects in a scene appear.

Quick Reference

Commands
DISTANTLIGHT
   Creates a distant light.
FREESPOT
   Creates free spotlight which is similar to a spotlight without a specified target.
LIGHT
   Creates a light.
POINTLIGHT
   Creates a point light that radiates light in all directions from its location.
SPOTLIGHT
   Creates a spotlight that emits a directional cone of light.
TARGETPOINT
   Creates a target point light.

System Variables
DEFAULTLIGHTING
   Turns on and off default lighting in place of other lighting.
DEFAULTLIGHTINGTYPE
   Specifies the type of default lighting, old or new.
LIGHTGLYPHDISPLAY
   Turns on and off the display of light glyphs.
LIGHTINGUNITS
Controls whether generic or photometric lights are used, and specifies the lighting units for the drawing.

LINEARBRIGHTNESS
Controls the brightness level of the viewport when using default lighting or generic lights.

LINEARCONTRAST
Controls the contrast level of the viewport when using default lighting or generic lights.

RENDERUSERLIGHTS
Controls whether to override the setting for viewport lighting during rendering.

Use Point Lights
A point light radiates light in all directions from its location.

Point Lights
A point light radiates light in all directions from its location. A point light does not target an object. Use point lights for general lighting effects. You can create a point light by entering the POINTLIGHT command or by selecting a point light from the Lights panel on the ribbon.

You create a target point light with the TARGETPOINT command. The difference between the target point light and a point light is the additional target properties that are available. A target light can be pointed to an object. A target point light can also be created from a point light by changing the target property of the point light from No to Yes.

In the standard lighting workflow, you can set a point light manually so its intensity diminishes with respect to distance either linearly, according to the inverse square of the distance, or not at all. By default, the attenuation is set to None.
Point Lights in Photometric Workflow

A free point light can have photometric distribution properties. The attenuation for a photometric point light is always set to inverse square.

When the LIGHTINGUNITS system variable is set to 1 (American units) or 2 (International SI units) for photometric lighting, additional properties are available for a point light. On the Properties Inspector palette, photometric properties are

- **Lamp Intensity.** Specifies the inherent brightness of the light. Specifies the intensity, flux or illuminance of the lamp.

- **Resulting Intensity.** Gives the final brightness of the light. (Product of lamp intensity and intensity factor. Read-only.)

- **Lamp Color.** Specifies the inherent color of the light in Kelvin temperature or standard.

- **Resulting Color.** Gives the final color of the light. This is determined by a combination of the lamp color and the filter color. (Product of lamp color and filter color. Read-only.)

**NOTE** When the drawing lighting units are photometric, the attenuation type property becomes disabled. Photometric lights have fixed, inverse-square attenuation.
The additional information about these properties are available under Lighting Properties.

**Quick Reference**

**Commands**

**LIGHT**
- Creates a light.

**POINTLIGHT**
- Creates a point light that radiates light in all directions from its location.

**TARGETPOINT**
- Creates a target point light.

**System Variables**

**DEFAULTLIGHTING**
- Turns on and off default lighting in place of other lighting.

**DEFAULTLIGHTINGTYPE**
- Specifies the type of default lighting, old or new.

**LIGHTGLYPHDISPLAY**
- Turns on and off the display of light glyphs.

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

**LIGHTSINBLOCKS**
- Controls whether lights contained in blocks are used when rendering.

**LINEARBRIGHTNESS**
- Controls the brightness level of the viewport when using default lighting or generic lights.

**LINEARCONTRAST**
- Controls the contrast level of the viewport when using default lighting or generic lights.
RENDERUSRERLIGHTS

Controls whether to override the setting for viewport lighting during rendering.

Use Spotlights

A spotlight can be directed towards an object.

Spotlights

A spotlight distribution casts a focused beam of light like a flashlight, a follow spot in a theater, or a headlight. A spotlight emits a directional cone of light. You can control the direction of the light and the size of the cone. Like a point light, a spotlight can be manually set to attenuate its intensity with distance. However, a spotlight's intensity will also always attenuate based on the angle relative to the spot's target vector. This attenuation is controlled by the hotspot and falloff angles of the spotlight. Spotlights are useful for highlighting specific features and areas in your model. A free spotlight (FREESPOT) is similar to spotlight. A spotlight has target properties.

Spotlights in Photometric Workflow

In photometric workflow, the hotspot intensity falls to 50 percent. The hotspot for standard lighting is at 100 percent. At its falloff angle, intensity of the spotlight falls to zero. Additional properties become available for a point light.
when LIGHTINGUNITS is set to 1 (American units) or 2 (International SI units)
for photometric lighting:

- **Lamp Intensity.** Specifies the inherent brightness of the light. Specifies the
  intensity, flux, or illuminance of the lamp.
- **Resulting Intensity.** Gives the final brightness of the light. (Product of lamp
  intensity and intensity factor. Read-only.)
- **Lamp Color.** Specifies the inherent color of the light in Kelvin temperature
  or standard.
- **Resulting Color.** Gives the final color of the light. This is determined by a
  combination of the lamp color and the filter color. (Product of lamp color
  and filter color. Read-only.)

**NOTE** When the drawing lighting units are photometric, the attenuation type
property becomes disabled. Photometric lights have fixed, inverse-square
attenuation. The hotspot falloff attenuation in the rendered image varies from
standard lighting, as it uses a different mathematical basis.

The additional information about these properties are available under Lighting
Properties.

**Quick Reference**

**Commands**

FREESPOT

 Creates free spotlight which is similar to a spotlight without a specified target.

LIGHT

 Creates a light.

SPOTLIGHT

 Creates a spotlight that emits a directional cone of light.

**System Variables**

DEFAULTLIGHTING

 Turns on and off default lighting in place of other lighting.
DEFAULTLIGHTINGTYPE

Specifies the type of default lighting, old or new.

LIGHTGLYPHDISPLAY

Turns on and off the display of light glyphs.

LIGHTINGUNITS

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

LIGHTSINBLOCKS

Controls whether lights contained in blocks are used when rendering.

LINEARBRIGHTNESS

Controls the brightness level of the viewport when using default lighting or generic lights.

LINEARCONTRAST

Controls the contrast level of the viewport when using default lighting or generic lights.

RENDERUSERLIGHTS

Controls whether to override the setting for viewport lighting during rendering.

Use Weblights

Weblights are photometric lights with customized, real-world light distributions.

Overview of Weblights

Weblights are photometric lights with customized, real-world light distributions.

A weblight (web) is a 3D representation of the light intensity distribution of a light source. Weblights can be used to represent anisotropic (non-uniform) light distributions derived from data provided by manufacturers of real-world lights. This gives a far more precise representation of the rendered light than either spot or point lights are capable of.
This directional light distribution information is stored in a photometric data file in the IES format using the IES LM-63-1991 standard file format for photometric data. You can load photometric data files provided by various manufacturers under the Photometric Web panel in the Properties Inspector palette for the light. The light icon represents the photometric web you select.

A light that uses a photometric web can be added to a drawing by entering the commands WEBLIGHT and FREEWEB at the command prompt. The WEBLIGHT command creates a targeted weblight, whereas the FREEWEB command creates a weblight without an explicit target.

To describe the directional distribution of the light emitted by a source, AutoCAD for Mac approximates the source by a point light placed at its photometric center. With this approximation, the distribution is characterized as a function of the outgoing direction only. The luminous intensity of the source for a predetermined set of horizontal and vertical angles is provided, and the system can compute the luminous intensity along an arbitrary direction by interpolation.

**NOTE** Web distribution is used only in rendered images. Weblights are approximated as point lights in the viewport.

**NOTE** You can control the size of a web glyph from Tools menu > Options > Drafting > Light Glyph Settings. Some web glyphs may appear very small on screen and may need to be adjusted.

**Goniometric Diagrams**

Photometric data is often depicted using a goniometric diagram.
Goniometric diagram of a web distribution
This type of diagram visually represents how the luminous intensity of a source varies with the vertical angle. However, the horizontal angle is fixed and, unless the distribution is axially symmetric, more than one goniometric diagram may be needed to describe the complete distribution.

Photometric Webs
The photometric web is a three dimensional representation of the light distribution. It extends the goniometric diagram to three dimensions, so that the dependencies of the luminous intensity on both the vertical and horizontal angles can be examined simultaneously. The center of the photometric web represents the center of the light object.

The luminous intensity in any given direction is proportional to the distance between this web and the photometric center, measured along a line leaving the center in the specified direction.

Example of Isotropic distribution
A sphere centered around the origin is a representation of an isotropic distribution. All the points in the diagram are equidistant from the center and therefore light is emitted equally in all directions.
Example of Ellipsoidal distribution

In this example, the points in the negative Z direction are the same distance from the origin as the corresponding points in the positive Z direction, so the same amount of light shines upward and downward. No point has a very large X or Y component, either positive or negative, so less light is cast laterally from the light source.

Quick Reference

Commands

LIGHT

Creates a light.

LIGHTLIST

Turns on and off the Lights in Model palette that lists all lights in the model.

FREEWEB

Creates a free web light which is similar to a web light without a specified target.
WEBLIGHT

Creates a web light.

**System Variables**

**DEFAULTLIGHTING**

Turns on and off default lighting in place of other lighting.

**DEFAULTLIGHTINGTYPE**

Specifies the type of default lighting, old or new.

**LIGHTGLYPHDISPLAY**

Turns on and off the display of light glyphs.

**LIGHTINGUNITS**

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

**LIGHTSINBLOCKS**

Controls whether lights contained in blocks are used when rendering.

**LINEARBRIGHTNESS**

Controls the brightness level of the viewport when using default lighting or generic lights.

**LINEARCONTRAST**

Controls the contrast level of the viewport when using default lighting or generic lights.

**RENDERUSERLIGHTS**

Controls whether to override the setting for viewport lighting during rendering.

**SUNSTATUS**

Turns on and off the lighting effects of the sun in the current viewport.

---

**IES Standard File Format**

IES standard file formats can be created and modified.

You can create a photometric data file in the IES format using the IES LM-63-1991 standard file format for photometric data. (IES stands for
Illuminating Engineering Society.) However, only the information relevant to AutoCAD for Mac is described here. For a complete description of the IES standard file format, see IES Standard File Format for Electronic Transfer of Photometric Data and Related Information, prepared by the IES Computer Committee (http://www.iesna.org).

The luminous intensity distribution (LID) of a luminaire is measured at the nodes of a photometric web for a fixed set of horizontal and vertical angles. The poles of the web lie along the vertical axis, with the nadir corresponding to a vertical angle of zero degrees. The horizontal axis corresponds to a horizontal angle of zero degrees and is oriented parallel to the length of the luminaire. This type of photometric web is generated by a Type C goniometer and is the most popular in North America; other types of goniometry are supported by the IES standard file format but are not discussed here.

The photometric data is stored in an ASCII file. Each line in the file must be less than 132 characters long and must be terminated by a carriage return/line-feed character sequence. Longer lines can be continued by inserting a carriage return/line-feed character sequence.

Each field in the file must begin on a new line and must appear exactly in the following sequence:

1  IESNA91
2  [TEST] The test report number of your data
3  [MANUFAC] The manufacturer of the luminaire
4  TILT=NONE
5  1
6  The initial rated lumens for the lamp used in the test or -1 if absolute photometry is used and the intensity values do not depend on different lamp ratings.
7  A multiplying factor for all the candela values in the file. This makes it possible to easily scale all the candela values in the file when the measuring device operates in unusual units—for example, when you obtain the photometric values from a catalog using a ruler on a goniometric diagram. Normally the multiplying factor is 1.
8  The number of vertical angles in the photometric web.
9  The number of horizontal angles in the photometric web.
10  1
11 The type of unit used to measure the dimensions of the luminous opening. Use 1 for feet or 2 for meters.

12 The width, length, and height of the luminous opening. It is normally given as 0 0 0.

13 1.0 1.0 0.0

14 The set of vertical angles, listed in increasing order. If the distribution lies completely in the bottom hemisphere, the first and last angles must be 0° and 90°, respectively. If the distribution lies completely in the top hemisphere, the first and last angles must be 90° and 180°, respectively. Otherwise, they must be 0° and 180°, respectively.

15 The set of horizontal angles, listed in increasing order. The first angle must be 0°. The last angle determines the degree of lateral symmetry displayed by the intensity distribution. If it is 0°, the distribution is axially symmetric. If it is 90°, the distribution is symmetric in each quadrant. If it is 180°, the distribution is symmetric about a vertical plane. If it is greater than 180° and less than or equal to 360°, the distribution exhibits no lateral symmetries. All other values are invalid.

16 The set of candela values. First, all the candela values corresponding to the first horizontal angle are listed, starting with the value corresponding to the smallest vertical angle and moving up the associated vertical plane. Then, the candela values corresponding to the vertical plane through the second horizontal angle are listed, and so on until the last horizontal angle. Each vertical slice of values must start on a new line. Long lines may be broken between values as needed by following the instructions given earlier.

**Example of Photometric Data File**

The following is an example of a photometric data file.
Quick Reference

Commands

DISTANTLIGHT

Creates a distant light.

FREESPOT

Creates free spotlight which is similar to a spotlight without a specified target.

FREEWEB

Creates a free web light which is similar to a web light without a specified target.

LIGHT

Creates a light.

POINTLIGHT

Creates a point light that radiates light in all directions from its location.

SPOTLIGHT

Creates a spotlight that emits a directional cone of light.

TARGETPOINT

Creates a target point light.

WEBLIGHT

Creates a web light.
System Variables

DEFAULTLIGHTING
  Turns on and off default lighting in place of other lighting.

DEFAULTLIGHTINGTYPE
  Specifies the type of default lighting, old or new.

LIGHTGLYPHDISPLAY
  Turns on and off the display of light glyphs.

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

LIGHTSINBLOCKS
  Controls whether lights contained in blocks are used when rendering.

LINEARBRIGHTNESS
  Controls the brightness level of the viewport when using default lighting or generic lights.

LINEARCONTRAST
  Controls the contrast level of the viewport when using default lighting or generic lights.

RENDERUSERLIGHTS
  Controls whether to override the setting for viewport lighting during rendering.

SUNSTATUS
  Turns on and off the lighting effects of the sun in the current viewport.

Common Lamp Values for Photometric Lights

Lists of commonly used lamps for defining photometric lights are available on the Lighting tool palette.

Commonly used lamps values are available on the Lighting tool palette: Fluorescent; Low Pressure Sodium; Incandescent, and High Intensity Discharge.
Quick Reference

Commands
FREEWEB
Creates a free web light which is similar to a web light without a specified target.
LIGHT
Creates a light.
WEBLIGHT
Creates a web light.

System Variables
LIGHTGLYPHDISPLAY
Turns on and off the display of light glyphs.
LIGHTINGUNITS
Controls whether generic or photometric lights are used, and specifies the lighting units for the drawing.
LIGHTSINBLOCKS
Controls whether lights contained in blocks are used when rendering.
LINEARBRIGHTNESS
Controls the brightness level of the viewport when using default lighting or generic lights.
LINEARCONTRAST
Controls the contrast level of the viewport when using default lighting or generic lights.
RENDERUSERLIGHTS
Controls whether to override the setting for viewport lighting during rendering.

Use Distant Lights
Distant lights are useful for lighting objects or as a backdrop.
Distant Lights in Standard Lighting Workflow

A distant light emits uniform parallel light rays in one direction only. You specify a FROM point and a TO point anywhere in the viewport to define the direction of the light. Spotlights and point lights are each represented by a different light glyph. Distant lights are not represented by glyphs in the drawing because they do not have a discrete position and affect the entire scene.

The intensity of a distant light does not diminish over distance; it is as bright at each face it strikes as it is at the source. Distant lights are useful for lighting objects or for lighting a backdrop uniformly.

**NOTE** It is recommended that you do not use distant lights in blocks.

Distant Lights in Photometric Workflow

Distant lights are not physically accurate. It is recommended that you do not use them in a photometric workflow.

Quick Reference

**Commands**

DISTANTLIGHT

Creates a distant light.

**System Variables**

LIGHTINGUNITS

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

Assigning a Shape to a Light

Assigning a shape to a light modifies the illumination of a scene.

**Area and Linear Lights**

The *Area* parameter on the light is a property of a light. Just as a light can have a color, it can also be assigned a shape. For example, you can shape it like a rectangle so that it acts like panel lighting in a ceiling. Or you can shape it like a line, so it acts like a narrow fluorescent light tube. The area light is a
way to assign a shape to the light. The shape affects the rendering and shadows in the same way that a panel light casts different light than a tube light in the real world.

**The Shape Property**

The Shape property is available in the Properties Inspector. Under ShadowDetail, there is a Type property. Whether the Shape property is displayed depends on the Type property that is selected.

The following types are available: Soft (shadow) map, Sharp (default), Soft (sampled). By selecting the Soft (sampled) option, the Shape property becomes available.

The available shapes depend on the type of light. You can select the Type property for the light distribution under the General panel in the Lighting category. If Spotlight and Soft (sampled) are selected, then the Shape types available are Rectangular and Disk. If Point and Soft (sampled) are selected, then the following shapes are available: Linear, Rectangular, Disk, Cylinder, and Sphere. If Web and Soft (sampled) are selected, then the following shapes are available: Linear, Rectangular, Disk, Cylinder, and Sphere. You can use the samples property on the area light to control the trade-off between rendering time and shadow accuracy.

The Visible in Render option is also displayed under ShadowDetail and controls the visibility of the shape when the scene is rendered.

**Adjust and Manipulate Lights**

You can add point lights, spotlights, and distant lights and set the location and properties of each.

**Control the Display of Lights**

The display of lights can be turned on and off in the drawing.

A light glyph is a graphic representation of a light. Point lights and spotlights can be placed in a drawing with a light glyph. Distant lights, such as sunlight, are not represented with a light glyph.

The display of lights can be controlled several ways. On the Command Line, the LIGHTGLYPHDISPLAY system variable controls the display of light glyphs in the drawing.
Displaying light glyphs in the plotted drawing is optional; light glyph display is controlled with the plot glyph property setting. With the plot glyph property, you specify light glyph to display one light at a time. The plot glyph setting for a viewport affects all the lights globally.

Quick Reference

**Commands**

DISTANTLIGHT
Creates a distant light.

LIGHT
Creates a light.

POINTLIGHT
Creates a point light that radiates light in all directions from its location.

SPOTLIGHT
Creates a spotlight that emits a directional cone of light.

**System Variables**

LIGHTGLYPHDISPLAY
Turns on and off the display of light glyphs.

Adjust Light Placement

After a light has been placed in a scene the position and target can be modified. The light, which is represented by a light glyph, can be repositioned after it is placed in the drawing. The light can be moved and rotated; the target can be modified. When the light glyph is selected, several grips are displayed.
NOTE  Rotating a targeted light is useful for aligning the area shadow region appropriately. Also, the orientation of the area shadow light is reset when the position or the target of the light is changed.

Location (Point Lights and Spotlights)
You can use the grip labeled Position to move a point light or a spotlight, or you can set the location in the Properties Inspector. The Position grip moves the light but does not change the target. To move both the light and its target, drag the light glyph itself.

Quick Reference

Commands
DISTANTLIGHT
 Creates a distant light.
FREESPOT
 Creates free spotlight which is similar to a spotlight without a specified target.
FREEWEB
 Creates a free web light which is similar to a web light without a specified target.
LIGHT
 Creates a light.
POINTLIGHT
Creates a point light that radiates light in all directions from its location.

SPOTLIGHT
Creates a spotlight that emits a directional cone of light.

TARGETPOINT
Creates a target point light.

WEBLIGHT
Creates a web light.

System Variables

DEFAULTLIGHTING
Turns on and off default lighting in place of other lighting.

DEFAULTLIGHTINGTYPE
Specifies the type of default lighting, old or new.

LIGHTGLYPHDISPLAY
Turns on and off the display of light glyphs.

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

LIGHTSINBLOCKS
Controls whether lights contained in blocks are used when rendering.

LINEARBRIGHTNESS
Controls the brightness level of the viewport when using default lighting or generic lights.

LINEARCONTRAST
Controls the contrast level of the viewport when using default lighting or generic lights.

RENDERUSERLIGHTS
Controls whether to override the setting for viewport lighting during rendering.
**SUNSTATUS**

Turns on and off the lighting effects of the sun in the current viewport.

---

**Control Light Properties**

Every light in the drawing has general and specific lighting properties that can be changed after the light is placed.

When a light is selected, its properties can be changed in the Properties Inspector.

You can use grip tools to move or rotate a selected light and change other properties such as the hotspot and falloff cone in spotlights. You can see the effect on the model as you change the properties of a light.

**General Properties**

The following properties are common to all lights. Full descriptions of the controls are located under the Properties command in Lighting Properties:

- **Name.** Specifies the name assigned to the light.
- **Type.** Specifies the type of light: point light, spotlight, distant light, or web.
- **On/Off Status.** Controls whether the light is turned on or off.
- **Shadows.** Controls whether the light casts shadows. To be displayed, shadows must be turned on in the visual style applied to the current viewport. Turn shadows off to increase performance.
- **Intensity factor.** Sets a multiplier that controls brightness. Intensity is not related to attenuation.
- **Filter color.** Sets the color of the light emitted.
- **Plot glyph.** Allows the ability to plot the drawing with the light glyphs on.

**Spotlight Hotspot and Falloff Under General Properties**

When light from a spotlight falls on a surface, the area of maximum illumination is surrounded by an area of lesser intensity.
- **Hotspot cone angle.** Defines the brightest part of a light beam. Also known as the *beam angle*.

- **Falloff cone angle.** Defines the full cone of light. Also known as the *field angle*.

- **Rapid decay area.** Consists of the region between the hotspot and falloff angles.

The greater the difference between the hotspot and falloff angles, the softer the edge of the light beam. If the hotspot and falloff angles are near equal, the edge of the light beam is sharp. Both values can range from 0 to 160 degrees. You can adjust these values directly with the Hotspot and Falloff grips.

**Photometric Properties**

Photometric lighting offers additional properties that make the lighting different than standard lighting. The following properties are under the Photometric properties panel:

- **Lamp intensity.** Specifies the inherent brightness of the light. Specifies the intensity, flux, or illuminance of the lamp.

- **Resulting intensity.** Gives the final brightness of the light. (Product of lamp intensity and intensity factor. Read-only.)

- **Lamp color.** Specifies the inherent color of the light in Kelvin temperature or standard.

- **Resulting color.** Gives the final color of the light. This is determined by a combination of the lamp color and the filter color. (Product of lamp color and filter color. Read-only.)
If you select Web in the Type property for a photometric light, additional properties are offered in the Photometric Web and Web offsets panel in the Lighting category.

- **Web file.** Specifies the data file describing the intensity distribution of the light.
- **Web preview.** Displays a 2D slice through goniometric data.
- **Rotation of X.** Specifies a rotational offset of the web about the optical X axis.
- **Rotation of Y.** Specifies a rotational offset of the web about the optical Y axis.
- **Rotation of Z.** Specifies a rotational offset of the web about the optical Z axis.

**Geometry Properties**

The Geometry properties control for the location and target point of the light. If the light is a target point light, spotlight, or weblight, additional target point properties are available. The Target property of a light can also be turned on or off.

**Attenuation Properties (Point Lights and Spotlights)**

Attenuation controls how light diminishes over distance. The farther away an object is from a light, the darker the object appears. You can specify no attenuation, inverse linear, or inverse squared (POINTLIGHT, SPOTLIGHT). Attenuation is not active for photometric lights.

- **None.** Sets no attenuation. Objects far from the point light are as bright as objects close to the light.
- **Inverse Linear.** Sets attenuation to be the inverse of the linear distance from the light. For example, at a distance of 2 units, light is half as strong as at the point light; at a distance of 4 units, light is one quarter as strong. The default value for inverse linear is half the maximum intensity.
- **Inverse Square.** Sets attenuation to be the inverse of the square of the distance from the light. For example, at a distance of 2 units, light is one quarter as strong as at the point light; at a distance of 4 units, light is one sixteenth as strong.
Another way to control the start point and end point of light is to use limits. Limits work like clipping planes to control where light is first emitted and where it stops. Using limits can increase performance by removing the need for the program to calculate light levels where the light is already practically invisible.

Quick Reference

Commands

DISTANTLIGHT
Creates a distant light.

FREESPOT
Creates free spotlight which is similar to a spotlight without a specified target.

FREEWEB
Creates a free web light which is similar to a web light without a specified target.

LIGHT
Creates a light.
POINTLIGHT
Creates a point light that radiates light in all directions from its location.

SPOTLIGHT
Creates a spotlight that emits a directional cone of light.

TARGETPOINT
Creates a target point light.

WEBLIGHT
Creates a web light.

**System Variables**

DEFAULTLIGHTING
Turns on and off default lighting in place of other lighting.

DEFAULTLIGHTINGTYPE
Specifies the type of default lighting, old or new.

LIGHTGLYPHDISPLAY
Turns on and off the display of light glyphs.

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

LIGHTSINBLOCKS
Controls whether lights contained in blocks are used when rendering.

LINEARBRIGHTNESS
Controls the brightness level of the viewport when using default lighting or generic lights.

LINEARCONTRAST
Controls the contrast level of the viewport when using default lighting or generic lights.

RENDERUSERLIGHTS
Controls whether to override the setting for viewport lighting during rendering.
SUNSTATUS
Turns on and off the lighting effects of the sun in the current viewport.

Incorporate Luminaire Objects
A luminaire object is a helper object that assembles a set of objects into a light fixture.

A luminaire object groups and manages the components of a light as a whole. Light fixtures can be represented by embedding photometric lights in blocks that also contain geometry. A luminaire object assembles a set of light objects into a light fixture.

An example of a luminaire object.

Quick Reference
System Variables
LIGHTSINBLOCKS
Controls whether lights contained in blocks are used when rendering.
Materials and Textures

Materials define the shininess, bumpiness, and transparency of object’s surfaces to give them a realistic appearance.

Overview of Materials

Add materials to objects in your drawings to provide a realistic effect in any rendered view.

Autodesk provides a large library of predefined materials for you to use. Use the Materials Browser to browse materials and apply them to objects in your drawing.

Textures add complexity and realism to a material. For example, to replicate the bumps in a paved road, you could apply a Noise texture to an object representing a road in a drawing. To replicate a brick and mortar pattern, you could use a Tile texture. Use the Texture Editor to define a texture’s appearance and how it is applied to an object.

Quick Reference

Commands
MATBROWSERCLOSE
Closes the Materials Browser.
MATBROWSEROPEN
Opens the Materials Browser.
MATERIALS
Opens the Materials Browser.
MATERIALSCLOSE
Closes the Materials Browser.

**System Variables**

CMATERIAL
Sets the material of new objects.

MATBROWSERSTATE
Controls the state of the Materials Browser.

**Browse Material Library**

You can browse and attach materials from the Materials Browser.

**Materials Browser**

Use the Materials Browser to navigate, sort, search, and select materials for use in your drawing.

The browser contains the following main components:

- **Browser toolbar.** Contains the Show or Hide Library Tree button and the search box.

- **Materials in the document.** Displays all the materials saved in the current drawing. You can sort the materials by name, type, swatch form, and color.

- **Material library tree.** Displays the Autodesk library.

- **Library details.** Displays previews of the materials in the selected categories.

**Materials Libraries**

The Autodesk library, with over 700 materials and over 1000 textures, is included with the product. The library is read-only, but you can copy Autodesk materials into the drawing, edit them, and save them to your own library.
Quick Reference

Commands
MATBROWSERCLOSE
Closes the Materials Browser.
MATBROWSEROPEN
Opens the Materials Browser.
MATERIALS
Opens the Materials Browser.
MATERIALSCLOSE
Closes the Materials Browser.

System Variables
MATBROWSERSTATE
Controls the state of the Materials Browser.
Render 3D Objects for Realism

A realistic rendering of a model can often give a product team or prospective client a clearer vision of a conceptual design than a plotted drawing.

Overview of Rendering

Rendering creates a 2D image based on your 3D scene. It shades the scene’s geometry using the lighting you’ve set up, the materials you’ve applied, and environmental settings such as background and fog.
The renderer is a general-purpose renderer that generates physically correct simulations of lighting effects, including ray-traced reflections and refractions, and global illumination.

A range of standard rendering presets, reusable rendering parameters, are available. Some of the presets are tailored for relatively quick preview renderings while others are for higher quality renderings.

Quick Reference

Commands

RENDER

- Creates a photorealistic or realistically shaded image of a 3D solid or surface model.

RENDERENVIRONMENT

- Controls visual cues for the apparent distance of objects.

RENDERWIN

- Displays the Render window without starting a rendering operation.

SAVEIMG

- Saves a rendered image to a file.

Prepare a Model for Rendering

The way a model is built plays an important role in optimizing rendering performance and image quality.

Understand Face Normals and Hidden Surfaces

There are several steps commonly taken to speed up the rendering process.

In order to minimize the time it takes to render a model, it is common practice to remove hidden surfaces or hide objects that are positioned off-camera. Furthermore, ensuring that all face normals orient in the same direction can also speed up the rendering process.

Every surface that you model is made up of faces. Faces are either triangular or quadrilateral and each face has an inward and outward oriented side. The
The direction in which a face is pointing is defined by a vector called a *normal*. The direction of the normal indicates the front, or outer surface of the face.

![Diagram of face normals](image)

When normals are unified and point in the same outward direction, the renderer processes each face and renders the model. If any normals are flipped, facing inward, the renderer skips them and leaves triangular or quadrilateral “holes” in the rendered image.

If a face is missing, you’ll need to manually reconstruct it. The direction of normals is determined by the way a face is drawn in a right-handed coordinate system: if you draw the face counter-clockwise, the normals point outward; if you draw the face clockwise, the normals point inward. You should draw faces consistently.

**NOTE** Solid objects have meshes and normals correctly oriented, which can be an aid to creating models for rendering.

![Diagram of consistent normals](image)

After the back faces have been removed, the renderer uses a Z buffer to compare relative distances along the Z axis. If the Z buffer indicates that one face overlaps another, the renderer removes the face that would be hidden.
The time saved is in proportion to the number of faces discarded out of the total number of faces.

Every object in a scene is processed by the renderer, even objects that are off camera and are not going to be present in the rendered view. A model that is built with the intent of rendering will benefit from good layer management. By turning off layers containing objects that are not in the view, you can increase rendering speed substantially.

See also:
- Use Models with Other Applications on page 1005
- Use a Visual Style to Display Your Model on page 92

**Minimize Intersecting and Coplanar Faces**

Certain kinds of geometry create special rendering problems.

The complexity of an object relates to the number of its vertices and faces. The more faces a model has, the longer it takes to render. Keep the geometry of your drawing simple to keep rendering time to a minimum. Use the fewest faces possible to describe a surface.

**Intersecting Faces**

Intersecting faces in a model occur when two objects pass through one other. For conceptual design situations, simply placing one object through another is a fast way to visualize how something will look. However, the edge created where the two objects intersect can exhibit a rippled appearance.

In the following example, the edge appears rippled in the left image and much cleaner after a Boolean union.
When edges do not appear to be as precise as you want, use Boolean operations like union, intersect, and subtract. A much cleaner and precise edge is created to better reflect the object’s appearance.

**Coplanar Faces**

Faces that overlap and lie in the same plane, *coplanar faces*, can produce ambiguous results, especially if the materials applied to the two faces differ.

In the following example, artifacts appear when faces occupy the same location.

Moving an object so its faces no longer occupy the same plane as another object will fix this situation.

**Twisted Faces**

Faces that self-overlap due to a 180-degree twist can also produce ambiguous results, because the normal for the face is not well defined.

In the following example, artifacts appear where the face is twisted due to crossing the second and third corner points.
This situation is often encountered when trying to fix a model that has a hole in its surface. For example, when corner points are selected for the new face, the points are crossed instead of being placed around the hole in a counter-clockwise direction. Avoid this problem by choosing corner points in the proper order.

**Quick Reference**

**Commands**

RENDER

Creates a photorealistic or realistically shaded image of a 3D solid or surface model.

**Balance Mesh Density for Smooth Geometry**

When you render a model, the density of the mesh affects the smoothness of surfaces.

Mesh components are comprised of vertices, faces, polygons, and edges.

- A vertex is a point that forms the corner of a face or polygon.
- A face is a triangular portion of a surface object.
- A polygon is a quadrilateral portion of a surface object.
- An edge is the boundary of a face or polygon.

In a drawing, all faces have three vertices, except faces in polyface meshes, which are treated as adjoining triangles. For rendering purposes, each quadrilateral face is a pair of triangular faces that share one edge.

Smoothing of an object is handled automatically by the renderer. Two types of smoothing occur during the rendering process. One smoothing operation interpolates the face normals across a surface. The other operation takes into account the number of faces, the face count, that make up the geometry; greater face counts result in smoother surfaces but longer processing times.

While you cannot control the interpolation of face normals, you can control the display accuracy of curved objects by using the VIEWRES command and the FACETRES system variable.
Control Display of Circles and Arcs

The VIEWRES command controls the display accuracy of curved 2D linework like circles and arcs in the current viewport.

In the following example, line segments are more apparent as VIEWRES decreases - Upper left = 1000, Middle = 100, Lower right = 10.

These objects are drawn on the screen using many short straight line segments. Smoother arcs and circles display with higher VIEWRES settings, but they take longer to regenerate. To increase performance while you're drawing, set a low VIEWRES value.

Control Display of Curved Solids

FACETRES controls the mesh density and smoothness of shaded and rendered curved solids.

In the following example, facets display on curved geometry when FACETRES is low. FACETRES = .25.

When FACETRES is set to 1, there is a one-to-one correlation between the viewing resolution of circles and arcs and the tessellation, a means of subdividing the faces of solid objects. For example, when FACETRES is set to 2, the tessellation will be twice the tessellation set by VIEWRES. The default value of FACETRES is 0.5. The range of possible values is 0.01 to 10.
When you raise and lower the value of VIEWRES, objects controlled by both VIEWRES and FACETRES are affected. When you raise and lower the value of FACETRES, only solid objects are affected.

In the following example, smoother geometry is displayed when FACETRES is set to higher values. FACETRES = 5.

See also:

■ Create Meshes on page 511

To alter the render resolution of solid geometry

1 At the Command prompt, enter facetres.

2 Do one of the following:
   ■ Enter a value greater than .5 to increase the smoothness of curved surfaces.
   ■ Enter a value lower than .5 to decrease the smoothness of curved surfaces.

To alter the display resolution of arcs and circles

1 At the Command prompt, enter viewres.

2 Ignore the prompt about fast zooms if you only want to make circles and arcs in the drawing look better for your rendering.

3 At the Circle Zoom Percent prompt, do one of the following:
   ■ Enter a value greater than 1000 to increase the smoothness of arcs and circles.
   ■ Enter a value lower than 1000 to decrease the smoothness of arcs and circles.
Quick Reference

Commands
VIEWRES
Sets the resolution for objects in the current viewport.

System Variables
FACETRES

Set Up the Renderer
You can control many of the settings that affect how the renderer processes a rendering task, especially when rendering higher quality images.

Control the Rendering Environment
You can use environmental features to set up atmospheric effects or background images.

You can enhance a rendered image by means of atmospheric effects like fog and depth cueing or by adding a bitmap image as a background.

Fog / Depth Cue Effects
Fog and depth cueing are very similar atmospheric effects that cause objects to appear to fade as they increase in distance from the camera. Fog uses a white color while depth cueing uses black.
The RENDERENVIRONMENT command is used to set up fog or depth cue parameters. The key parameters you’ll set are the color of the fog or depth cueing, the near and far distances, and the near and far fog percentages.

Fog and depth cueing are based on the front or back clipping planes of your camera coupled with the near and far distance settings on the Render Environment dialog box. For example, the back clipping plane of a camera is active and located 30 feet from the camera location. If you want fog to start 15 feet from the camera and spread away indefinitely, you set the Near Distance to 50 and the Far Distance to 100.

The density of the fog or depth cueing is controlled by the Near and Far Fog Percentages. These settings have a range of 0.0001 to 100. Higher values mean the fog or depth cueing is more opaque.

**TIP** For smaller scale models, the Near and Far Fog Percentage setting may need to be set below 1.0 to see the desired effect.

**Quick Reference**

**Commands**

RENDERENVIRONMENT

Controls visual cues for the apparent distance of objects.

**Basics of Rendering**

While the final goal is to create a photorealistic, presentation-quality image that illustrates your vision, you create many renderings before you reach that goal.

At a basic level, you can use the RENDER command to render your model without applying any materials, adding any lights, or setting up a scene. When you render a new model, the renderer automatically uses a virtual “over-the-shoulder” distant light. You cannot move or adjust this light.

**Render Views**

You render the view displayed in the current viewport.

When rendering, all objects in the current viewport are rendered. If you have not set a named view current, the current view is rendered. While the rendering
process is faster when you render smaller portions of a view, rendering the entire view lets you see how all objects are oriented to one another.

If your current drawing contains named views, you can quickly display them by using the VIEW command.

The following example shows a rendering of a named view.

For a complete description of the Render Window, see RENDER in the Command Reference.

Quick Reference

Commands

RENDER

Creates a photorealistic or realistically shaded image of a 3D solid or surface model.

Save and Redisplay Rendered Images

You can save a rendering and then redisplay it later. Redisplaying is much faster than rendering again.

Save a Rendered Image

After creating a rendering, you can save the image to for redisplay at a later time.
Depending on the render settings and render preset you’ve chosen, rendering can be a time-consuming process. However, redisplaying a previously rendered image is instantaneous.

Once rendering is complete, you can save the image or save a copy of the image to one of the following file formats: BMP, TGA, TIF, PCX, JPG, or PNG.

Quick Reference

Commands

RENDER

Creates a photorealistic or realistically shaded image of a 3D solid or surface model.

RENDERWIN

Displays the Render window without starting a rendering operation.

SAVEIMG

Saves a rendered image to a file.

Redisplay a Rendered Image

Having saved your rendered image, you can redisplay that rendering at any time.

If you don’t want to use an external program, you can also view the image by inserting it into the drawing with the IMAGEATTACH command.

Quick Reference

Commands

IMAGEATTACH

Inserts a reference to an image file.

RENDER

Creates a photorealistic or realistically shaded image of a 3D solid or surface model.
Use Models with Other Applications

You can use other Autodesk products to further enhance the quality of your models.

Autodesk® products continually improve their ability to share drawings and models, including 3ds Max®, Autodesk® VIZ, and AutoCAD® Architecture.

3ds Max or Autodesk VIZ

With these products, you can make greater improvements on your models. You have the option of opening DWG or DXF files without converting or you can use the File Link Manager to create a live link with a drawing file. 3ds Max or Autodesk VIZ offers expanded animation, lighting, material, and rendering capabilities that add further polish to your presentation graphics.

AutoCAD Architecture

AutoCAD for Mac geometry can be opened in AutoCAD Architecture, no conversion is necessary. Once the model is open you can use architectural related features, such as architectural objects, schedules or integrated rendering, to streamline your architectural design and documentation process.
Commands associated with definitions are shown in parentheses at the end of the definition.

A CUIx file that is typically controlled by a CAD manager. It is often accessed by many users and is stored in a shared network location. The file is read-only to users to prevent the data in the file from being changed. A CAD manager creates an enterprise CUIx file by modifying a main CUIx file and then saving the file to the support location defined in the Options dialog box, Files tab.

3D mesh primitive Basic mesh forms such as boxes, cones, cylinders, pyramids, wedges, spheres, and tori.

3D view Any view where the UCS icon appears in rendered colored form; current visual style is not 2D Wireframe, and the model is being viewed from an isometric view.

absolute coordinates Coordinate values measured from a coordinate system's origin point. See also origin, relative coordinates, user coordinate system (UCS), world coordinates, and world coordinate system (WCS).

acquired point In the tracking or object snap tracking methods of locating a point, an intermediate location used as a reference.

acquisition marker During tracking or object snap tracking, the temporary plus sign displayed at the location of an acquired point.

Action bar Toolbar-like UI that displays the actions associated with a parameter object.

activate Part of the Autodesk software registration process. It allows you to run a product in compliance with the product's end-user license agreement.

adaptive degradation A method of controlling performance that turns off features in a certain order when performance falls below a specified level.

adaptive sampling A method to accelerate the anti-aliasing process within the bounds of the sample matrix size. See also anti-aliasing.
adjacent cell selection A selection of table cells that share at least one boundary with another cell in the same selection.

alias A shortcut for a command. For example, CP is an alias for COPY, and Z is an alias for ZOOM. You define aliases in the acad.pgp file.

aliasing The effect of discrete picture elements, or pixels, aligned as a straight or curved edge on a fixed grid appearing to be jagged or stepped. See also anti-aliasing.

aligned dimension A dimension that measures the distance between two points at any angle. The dimension line is parallel to the line connecting the dimension’s definition points. (DIMALIGNED)

alpha channel Alpha is a type of data, found in 32-bit bitmap files, that assigns transparency to the pixels in the image.

A 24-bit truecolor file contains three channels of color information: red, green, and blue, or RGB. Each channel of a truecolor bitmap file is defined by 8 bits, providing 256 levels of intensity. The intensity of each channel determines the color of the pixel in the image. Thus, an RGB file is 24-bit with 256 levels each of red, green, and blue.

By adding a fourth, alpha channel, the file can specify the transparency, or opacity, of each of the pixels. An alpha value of 0 is transparent, an alpha value of 255 is opaque, and values in between are semi-transparent. An RGBA file (red, green, blue, alpha) is 32-bit, with the extra 8 bits of alpha providing 256 levels of transparency.

To output a rendered image with alpha, save in an alpha-compatible format such as PNG, TIFF, or Targa.

ambient color A color produced only by ambient light. Ambient color is the color of an object where it is in shadow. This color is what the object reflects when illuminated by ambient light rather than direct light.

ambient light Light that illuminates all surfaces of a model with equal intensity. Ambient light has no single source or direction and does not diminish in intensity over distance.

angular dimension A dimension that measures angles or arc segments and consists of text, extension lines, and leaders. (DIMANGULAR)

angular unit The unit of measurement for an angle. Angular units can be measured in decimal degrees, degrees/minutes/seconds, grads, and radians.

annotation scale A setting that is saved with model space, layout viewports, and model views. When you create annotative objects, they are scaled based
on the current annotation scale setting and automatically displayed at the correct size.

**annotational constraint** Dimensional constraint used to control the size of the geometry as well as annotate the drawing.

*See also parameter constraint, and dynamic constraint*

**annotations** Text, dimensions, tolerances, symbols, notes, and other types of explanatory symbols or objects that are used to add information to your model.

**annotative** A property that belongs to objects that are commonly used to annotate drawings. This property allows you to automate the process of scaling annotations. Annotative objects are defined at a paper height and display in layout viewports and model space at the size determined by the annotation scale set for those spaces.

**anonymous block** An unnamed block created by a number of features, including associative and nonassociative dimensions.

**anti-aliasing** A method that reduces aliasing by shading the pixels adjacent to the main pixels that define a line or boundary. *See also aliasing.*

**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.

![Sample arrowheads](image)

**aspect ratio** Ratio of display width to height.
associative dimension A dimension that automatically adapts as the associated geometry is modified. Controlled by the DIMASSOC system variable. See also nonassociative dimension and exploded dimension.

associative hatch Hatching that conforms to its bounding objects such that modifying the bounding objects automatically adjusts the hatch. (BHATCH)

associative surfaces Associative surfaces automatically adjust their location and shape when the geometric objects associated with them are modified. Controlled by the SURFACEASSOCIATIVITY system variable.

attenuation The diminishing of light intensity over distance.

attribute definition An object that is included in a block definition to store alphanumeric data. Attribute values can be predefined or specified when the block is inserted. Attribute data can be extracted from a drawing and inserted into external files. (ATTDEF)

attribute extraction file A text file to which extracted attribute data is written. The contents and format are determined by the attribute extraction template file. See also attribute extraction template file.

attribute extraction template file A text file that determines which attributes are extracted and how they are formatted when written to an attribute extraction file. See also attribute extraction file.

attribute prompt The text string displayed when you insert a block with an attribute whose value is undefined. See also attribute definition, attribute tag, and attribute value.

attribute tag A text string associated with an attribute that identifies a particular attribute during extraction from the drawing database. See also attribute definition, attribute prompt, and attribute value.

attribute value The alphanumeric information associated with an attribute tag. See also attribute definition, attribute prompt, and attribute tag.

AutoCAD for Mac library search path

axis tripod Icon with X, Y, and Z coordinates that is used to visualize the viewpoint (view direction) of a drawing without displaying the drawing. (VPOINT)

B-spline curve A blended piecewise polynomial curve passing near a given set of control points. See also Bezier curve. (SPLINE)

back face The opposite side of a front face. Back faces are not visible in a rendered image. See also front faces.
**base point** 1. In the context of editing grips, the grip that changes to a solid color when selected to specify the focus of the subsequent editing operation. 2. A point for relative distance and angle when copying, moving, and rotating objects. 3. The insertion base point of the current drawing. (BASE) 4. The insertion base point for a block definition. (BLOCK)

**baseline** An imaginary line on which text characters appear to rest. Individual characters can have descenders that drop below the baseline. See also baseline dimension.

**baseline dimension** Multiple dimensions measured from the same baseline. Also called parallel dimensions. See also baseline.

**basic tooltip** Displays a brief description for the tooltip.

**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 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)

**bounded area** A closed area that consists of a single object (such as a circle) or of multiple, coplanar objects that overlap. You can insert hatch fills within bounded areas. Bounded areas are also used to create 3D objects through extrusion by using the PRESSPULL command.
bump map A map in which brightness values are translated into apparent changes in the height of the surface of an object.

BYBLOCK A special object property used to specify that the object inherits the color or linetype of any block containing it. See also BYLAYER.

BYLAYER A special object property used to specify that the object inherits the color or linetype associated with its layer. See also BYBLOCK.

callout block A block used as symbol to reference another sheet. Callout blocks have many industry-specific terms, such as reference tags, detail keys, detail markers, and so on. See also label block.

camera Defines the current eye-level position in a 3D model. A camera has a location XYZ coordinate, a target XYZ coordinate, and a field of view or lens length, which determines the magnification or zoom factor.

camera target Defines the point you are viewing by specifying the coordinate at the center of the view.

candela The SI unit of luminous intensity (perceived power emitted by a light source in a particular direction) (Symbol: cd). Cd/Sr

category See view category.

cell The smallest available table selection.

cell boundary The four gridlines surrounding a table cell. An adjacent cell selection can be surrounded with a cell boundary.

cell style A style that contains specific formatting for table cells.

circular external reference An externally referenced drawing (xref) that references itself directly or indirectly. The xref that creates the circular condition is ignored.

clipping planes The boundaries that define or clip the field of view.

CMYK For cyan, magenta, yellow, and key color. A system of defining colors by specifying the percentages of cyan, magenta, yellow, and the key color, which is typically black.

Color bleed scale Increases or decreases the saturation of the reflected color from the material.

color map A table defining the intensity of red, green, and blue (RGB) for each displayed color.

column A vertically adjacent table cell selection spanning the height of the table. A single column is one cell in width.
command line  A text area reserved for keyboard input, prompts, and messages.

compass  A visual aid that indicates the directions North, South, East, and West in the current model.

composite solid  A solid created from two or more individual solids. (UNION, SUBTRACT, INTERSECT)

constraint bar Displays the geometric constraints associated with objects or with points on objects.

constraint point  Point on an object that can be geometrically and/or dimensionally constrained (for example, an endpoint or an insertion point).

constraints  Form of parametric design. Rules that govern the position, slope, tangency, dimensions, and relationships among objects in a geometry.

construction plane  See workplane.

continued dimension  A type of linear dimension that uses the second extension line origin of a selected dimension as its first extension line origin, breaking one long dimension into shorter segments that add up to the total measurement. Also called chain dimension. (DIMCONTINUE)

control frame  A series of point locations used as a mechanism to control the shape of a B-spline. These points are connected by a series of line segments for visual clarity and to distinguish the control frame from fit points. The CVSHOW and CVHIDE commands must be turned on to display and hide control frames.

control point  See control frame.

Coons patch  In 3D surface meshes, the bicubic surface (one curved in the M direction and another in the N direction) interpolated between four edges.

coordinate filters  Functions that extract individual X, Y, and Z coordinate values from different points to create a new, composite point. Also called X,Y,Z point filters.

crease  A sharpened ridge that defines one or more edges of a mesh face subobject. (MESHCREASE)

cross sections  Generally, curves or lines that define the profile (shape) of a lofted solid or surface. Cross sections can be open or closed. A lofted solid or surface is drawn in the space between the cross sections. (LOFT)

crosshairs  A type of cursor consisting of two lines that intersect.
crossing selection A rectangular area drawn to select objects fully or partly within its borders.

CTB file SA color-dependent plot style table.

ctrl-cycle Method for cycling between different behaviors while editing geometry, either in a command or when grip-editing. Pressing and releasing the Ctrl key cycles the behavior. For constrained geometry, Ctrl-cycling switches between enforcing and relaxing constraints.

current drawing A drawing file that is open in the program, and receives any command or action that you enter.

cursor See pointer and crosshairs.

cursor menu See shortcut menu.

curve-fit A smooth curve consisting of arcs joining each pair of vertices. The curve passes through all vertices of the polyline and uses any tangent direction you specify.

custom grips In a dynamic block reference, used to manipulate the geometry and custom properties.

custom object A type of object that is created by an ObjectARX application and that typically has more specialized capabilities than standard objects. Custom objects include parametric solids (AutoCAD Mechanical Desktop), intelligently interactive door symbols (AutoCAD Architecture), polygon objects (AutoCAD Map 3D), and associative dimension objects (AutoCAD and AutoCAD LT). See also proxy object and object enabler.

customization (CUI) file An XML-based file that stores customization data for the user interface. You modify a customization file through the Customize dialog box.

decimal degrees A notation for specifying latitude and longitude. For example, 35.1234°, 100.5678°. Latitude always precedes longitude

default drawing See initial environment.

default lighting The lighting in a shaded viewport when the sun and user lights are turned off. Faces are lighted by two distant light sources that follow the viewpoint as you move around the model.
**default value** The value that is accepted when you press Enter at a sub-prompt. The default value is displayed in angle brackets <>.

**See also** default.

**definition points** Points for creating a dimension. The program refers to the points to modify the appearance and value of a nonassociative dimension when the dimensioned object is modified. Also called *defpoints* and stored on the special layer DEFPOINTS.

**definition table** The nongraphical data area of a drawing file that stores block definitions.

**dependency highlighting** In a dynamic block definition, how associated objects are displayed when you select a parameter, grip, or action.

**dependent named objects (in xrefs)** Named objects brought into a drawing by an external reference. See also named object and symbol table.

**dependent symbols** See dependent named objects (in xrefs).

**DIESEL** For *Direct Interpretively Evaluated String Expression Language*.

**diffuse color** An object’s predominant color.

**dimension line arc** An arc (usually with arrows at each end) spanning the angle formed by the extension lines of an angle being measured. The dimension text near this arc sometimes divides it into two arcs. See also *angular dimension*.

**dimension style** A named group of dimension settings that determines the appearance of the dimension and simplifies the setting of dimension system variables. (DIMSTYLE)

**dimension text** The measurement value of dimensioned objects.

**dimension variables** A set of numeric values, text strings, and settings that control dimensioning features. (DIMSTYLE)

**dimensional constraint** Parametric dimensions that control the size, angle, or position of geometry relative to the drawing or other objects. When dimensions are changed, the object resizes.

**direct distance entry** A method to specify a second point by first moving the cursor to indicate direction and then entering a distance.

**dithering** Combining color dots to give the impression of displaying more colors than are actually available.

**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 for...
Mac window and on how many toolbars and other elements are displayed. 
See also AutoCAD for Mac 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 template A drawing file with preestablished settings for new drawings such as acad.dwt and acadiso.dwt however, any drawing can be used as a template. See also initial environment.

driven constraint A non-parametric dimension enclosed in parentheses that shows the current value of geometry. The value is updated when the geometry changes size, but it does not control geometry.

driving dimension A parametric dimension that determines the size of geometry and resizes the object when its value changes.

driving property A lookup property is considered invertible when a manual change in the lookup value for a block reference causes other properties values change.

DWG Standard file format for saving vector graphics.

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.

dynamic constraint Dimensional constraint (Constraint Form property = "dynamic") that displays the constraints only when you select the constrained object.

See also: parameter constraint
See also: annotational constraint

dynamic dimension Temporary dimensions that appear on objects, including dynamic block references, when they are grip edited.
edge The boundary of a face.

edge modifiers Effects such as overhang and jitter that control how edges are displayed in a shaded model.

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)

empty selection set A selection set that contains no objects.

environment map A bitmap that is used to simulate reflections in materials that have reflective properties. The map is “wrapped” around the scene and any reflective object will show the appropriate portion of the map in the reflective parts of its material.

environment variable A setting stored in the operating system that controls the operation of a program.

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.

face color mode A setting in the visual style that controls how color is displayed on a face.
**face style** A setting in the visual style that defines the shading on a face.

**facet** The underlying structure of the face of a 3D solid, surface, or mesh. Facets can be quadrilateral or triangular. Smoothing a mesh object increases the number of facets for each face.

**feature control frame** The tolerance that applies to specific features or patterns of features. Feature control frames always contain at least a geometric characteristic symbol to indicate the type of control and a tolerance value to indicate the amount of acceptable variation.

**fence** A multisegmented line specified to select objects it passes through.

**field** A specialized text object set up to display data that may change during the life cycle of the drawing. When the field is updated, the latest value of the field is displayed. (FIELD)

**fill** A solid color covering an area bounded by lines or curves. (FILL)

**filters** See coordinate filters.

**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, made up of letters, numbers, punctuation marks, and symbols of a distinctive proportion and design.

**footcandle** The American unit of illuminance (symbol: fc). Lm/ft^2.

**footcandle** The American unit of illuminance (symbol: fc). Lm/ft^2

**freeze** A setting that suppresses the display of objects on selected layers. Objects on frozen layers are not displayed, regenerated, or plotted. Freezing layers shortens regenerating time. See also thaw. (LAYER)

**front faces** Faces with their normals pointed outward.

**general property** Properties that are common between a selection of objects. These include Color, Layer, Linetype, Linetype scale, Plot style, Lineweight, Transparency, and Thickness.

**generic surface** A 3D surface object with no control vertices, history, or analytic information. Generic surfaces cannot be associative and they are created when associative analytic surfaces are separated or by using the BREP command. See also procedural surface and NURBS surface.
geometric constraint  Rules that define the geometric relationships of objects (or points of objects) elements and control how an object can change shape or size.

Geometric constraints are coincident, collinear, concentric, equal, fix, horizontal, parallel, perpendicular, tangent, and vertical.

gemetry  All graphical objects such as lines, circles, arcs, polylines, and dimensions. Nongraphical objects, such as linetypes, lineweights, text styles, and layers are not considered geometry. See also named object.

gizmo  A tool that permits you to manipulate a 3D object uniformly or along a specified axis or plane. Examples of gizmos include the 3D Move, 3D Rotate, and 3D Scale gizmos. They are displayed when you select a 3D object.

global illumination  An indirect illumination technique that allows for effects such as color bleeding. As light hits a colored object in the model, photons bounce to adjacent objects and tint them with the color of the original object.

Gooch shading  A type of shading that uses a transition from cool to warm colors rather than from dark to light.

graphics area  See drawing area.

grid  An area covered with regularly spaced dots or lines to aid drawing. The grid spacing is adjustable. The grid dots are never plotted. See also grid limits. (GRID)

grid limits  The user-defined rectangular boundary of the drawing area covered by dots when the grid is turned on. Also called drawing limits. (LIMITS)

grip modes  The editing capabilities activated when grips are displayed on an object: stretching, moving, rotating, scaling, and mirroring.

grip tool  An icon that you use in a 3D view to easily constrain the movement or rotation of a selection set of objects to an axis or a plane. (3DMOVE, 3DROTATE)
grips Small squares and triangles that appear on objects you select. After selecting the grip, you edit the object by dragging it with the pointing device instead of entering commands.

ground plane The XY plane of the user coordinate system when perspective projection is turned on. The ground plane displays with a color gradient between the ground horizon (nearest to the horizon) and the ground origin (opposite the horizon). See also sky and underground.

guide curves Lines or curves that intersect each cross section of a lofted solid or surface and that define the form by adding additional wireframe information to the object. (LOFT)

handle A unique alphanumeric tag for an object in the program's database.

heads-up display (HUD) The process of transparently displaying user interface elements on top of or over the drawing area without obscuring the view of the objects drawn on the drawing area.

helix An open 2D or 3D spiral. (HELIX)

Help menu In the AutoCAD for Mac, you can find Help on the Mac IOS menu bar or by pressing Fn-F1.

HLS For hue, lightness, and saturation. A system of defining color by specifying the amount of hue, lightness, and saturation.

Home view A special view saved with the drawing that is controlled through the ViewCube tool. The Home view is similar in concept to the default, initial view presented when a drawing is first opened.

horizontal landing An optional line segment connecting the tail of a leader line with the leader content.

Illuminance In photometry, illuminance is the total luminous flux incident on a surface per unit area.

indirect bump scale Scales the effect of the base material's bump mapping in areas lit by indirect light.

indirect illumination Illumination techniques such as global illumination and final gathering, that enhance the realism of a scene by simulating radiosity, or the interreflection of light between objects in a scene.

initial environment The variables and settings for new drawings as defined by the default drawing template, such as acad.dwt or acadiso.dwt. See also template drawing.
interface element  A user interface object that can be customized, such as a pull-down menu or tool set.

interpolation points  Defining points that a B-spline passes through. See also approximation points and fit points.

island  An enclosed area within another enclosed area. Islands may be detected as part of the process of creating hatches, polylines, and regions. (BHATCH, BOUNDARY)

ISO  For International Standards Organization. The organization that sets international standards in all fields except electrical and electronics. Headquarters are in Geneva, Switzerland.

isometric snap style  A drafting option that aligns the cursor with two of three isometric axes and displays grid, making 2D isometric drawings easier to create.

landing  The portion of a leader object that acts as a pointer to the object being called out. A landing can either be a straight line or a spline curve.

landing gap  An optional space between a leader tail and the leader content.

layer  A logical grouping of data that are like transparent acetate overlays on a drawing. You can view layers individually or in combination. (LAYER)

layout  The environment in which you create and design paper space layout viewports to be plotted. Multiple layouts can be created for each drawing.

layout viewports  Objects that are created in paper space that display views. See also paper space. (VPORTS)

leader tail  The portion of a leader line that is connected to the annotation.

lens length  Defines the magnification properties of a camera’s lens. The greater the lens length, the narrower the field of view.

level of smoothness  The property assigned to a mesh object to control how much the edges of the object are smoothed. Level 0 (zero) represents the least rounded shape for a specified mesh object. Higher levels result in increased smoothness.

light glyph  The graphic representation of a point light or a spotlight.

limits  See drawing limits.

line font  See linetype.

linetype  How a line or type of curve is displayed. For example, a continuous line has a different linetype than a dashed line. Also called line font. (LINETYPE)
lineweight A width value that can be assigned to all graphical objects except TrueType® fonts and raster images.

**LL84 coordinate system** Common latitude longitudinal-based coordinate system where latitude and longitude are both measured from -90 to 90 degrees. Longitude begins at 0 degrees at the Prime Meridian in Greenwich, England and is measured from -180 to 180. Latitude is 0 degrees at the equator and is measured from -90 to 90.

**lofted solid/surface** A solid or surface that is drawn through a set of two or more cross-section curves. The cross sections define the profile (shape) of the resulting solid or surface. Cross sections (generally, curves or lines) can be open or closed. (LOFT)

**lumen** The SI unit of luminous flux (Symbol: lm). Cd * Sr

**luminaire** This refers to the aggregation of a lamp or lamps and its fixture. The fixture may be a simple can or a complex armature with constrained joints.

**luminance** Luminance is the value of light reflected off a surface. It is a measure of how bright or dark we perceive the surface.

**luminous flux** The perceived power in per unit of solid angle. The total luminous flux for a lamp is the perceived power emitted in all directions.

**lux** The SI unit of illuminance (symbol: lx). Lm/m^2

**main customization file** A writable CUI file that defines most of the user interface elements (including the pull-down menus and tool sets).

**merge** In tables, an adjacent cell selection that has been combined into a single cell.

**mesh** A tessellated, or subdivided object type that is defined by faces, edges, and vertices. Mesh can be smoothed to achieve a more rounded appearance and creased to introduce ridges. Before AutoCAD 2010, only the less modifiable polygon and polyface mesh was available.

**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)

**multileader** A leader object that creates annotations with multiple leader lines.

**named object** Describes the various types of nongraphical information, such as styles and definitions, stored with a drawing. Named objects include linetypes, layers, dimension styles, text styles, block definitions, layouts, views, and viewport configurations. Named objects are stored in definition (symbol) tables.

**named objects, dependent** See dependent named objects (in xrefs).

**named view** A view saved for restoration later. (VIEW)

**node** An object snap specification to locate points, dimension definition points, and dimension text origins.

**non-associative dimension** A dimension that does not automatically change as the associated geometry is modified. Controlled by the DIMASSOC system variable. See also associative dimension and exploded dimension.

**normal** A normal is a vector that defines which way a face is pointing. The direction of the normal indicates the front, or outer surface of the face.

**noun-verb selection** Selecting an object first and then performing an operation on it rather than entering a command first and then selecting the object.

**NURBS** For nonuniform rational B-spline curve. A B-spline curve or surface defined by a series of weighted control points and one or more knot vectors. See also B-spline curve.

**NURBS surface** Surfaces that have control vertices in the U and V directions. NURBS surfaces cannot be associative. See also procedural surface and generic surface.

**object** One or more graphical elements, such as text, dimensions, lines, circles, or polylines, treated as a single element for creation, manipulation, and modification. Formerly called entity.

**Object 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 for Mac Runtime Extension) A compiled-language programming environment for developing AutoCAD for Mac applications.

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.

palette A user interface element that can be either docked, anchored, or floating in the drawing area. Dockable windows include the command line, status bar, Properties Inspector, and so on.

pan To shift the view of a drawing without changing magnification. See also zoom. (PAN)

paper space One of two primary spaces in which objects reside. Paper space is used for creating a finished layout for printing or plotting, as opposed to doing drafting or design work. You design your model using the Model tab. See also model space and viewport. (PSPACE)

parametric design Ability to establish relationships between objects, to drive the size and orientation of geometry with model and user-defined parameters.

parametric drawing Feature in AutoCAD that assigns constraints to objects, establishing the distance, location, and orientation of objects with respect to other objects.

path curve Defines the direction and length that a profile curve is lofted, swept, or extruded to create a solid or surface. (Sweep, Loft, Extrude)

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.

**perspective view** Objects in 3D seen by an observer positioned at the
viewpoint looking at the view center. Objects appear smaller when the distance
from the observer (at the view point) to the view center increases. Although
a perspective view appears realistic, it does not preserve the shapes of objects.
Parallel lines seemingly converge in the view. The program has perspective
view settings for VPORTS table entries as well as viewport objects.

**photometric lights** Photometric lights are physically-correct lights. Physically
correct lights attenuate as the square of the distance. Photometry is the science
of measurement of visible light in terms of its perceived brightness.

**photorealistic rendering** Rendering that resembles a photograph.

**pick button** The button on a pointing device that is used to select objects or
specify points on the screen. For example, on a two-button mouse, it is the
left button by default.

**pick points** Clicking and acquiring a point on an object in the drawing.

**plan view** A view orientation from a point on the positive Z axis toward the
origin (0,0,0). (PLAN)

![plan view diagram]

**planar face** A flat face that can be located anywhere in 3D space.

**planar projection** Mapping of objects or images onto a plane.

**planar surface** A flat surface that can be located anywhere in 3D space.
(PLANESURF)

**pline** See polyline.

**plot style** An object property that specifies a set of overrides for color,
dithering, gray scale, pen assignments, screening, linetype, lineweight,
endstyles, joinstyles, and fill styles. Plot styles are applied at plot time.
plot style table  A set of plot styles. Plot styles are defined in plot style tables and apply to objects only when the plot style table is attached to a layout or viewport.

PMP file  Plot Model Parameter. File containing custom plotter calibration and custom paper size information associated with plotter configuration file.

point  1. A location in three-dimensional space specified by $X$, $Y$, and $Z$ coordinate values. 2. An object consisting of a single coordinate location. (POINT)

point filters  See coordinate filters.

pointer  A cursor on a video display screen that can be moved around to place textual or graphical information. See also crosshairs.

polar array  Objects copied around a specified center point a specified number of times. (ARRAY)

Polar Snap  A precision drawing tool used to snap to incremental distances along the polar tracking alignment path. See also polar tracking on page 1026.

polar tracking  A precision drawing tool that displays temporary alignment paths defined by user-specified polar angles. See also Polar Snap.

polyface and polygon mesh  Legacy mesh types that were available before AutoCAD 2010. Although you can continue to create polygonal and polyface mesh (for example, by setting MESHTYPE to 0), the newer, more modifiable mesh type is recommended.

polygon window selection  A multisided area specified to select objects in groups. See also crossing selection and window selection.

polyline  An object composed of one or more connected line segments or circular arcs treated as a single object. Also called pline. (PLINE, PEDIT)

polysolid  A swept solid that is drawn the same way you draw a polyline or that is based on an existing line. By default, a polysolid always has a rectangular profile. You can specify the height and width of the profile. (POLYSOLID)

pre-selection set  A selection set of objects that is defined prior to the execution of an action macro.

primary table fragment  The fragment of a broken table that contains the beginning set of rows up to the first table break.

primitive  Basic 3D forms such as boxes, cones, cylinders, pyramids, wedges, spheres, and tori. You can create primitive meshes and primitive 3D solid objects.
**procedural materials** Materials that generate a 3D pattern in two or more colors, and apply it to an object. These include marble and wood. Also called *template materials*.

**procedural surface** A 3D surface object that has history and analytic information, but no control vertices. Procedural surfaces are the only type of surface that can be associative. See also *generic surface* and *NURBS surface*.

**profile curve** An object that is swept, extruded, or revolved and defines the shape of the resulting solid or surface. (SWEEP, EXTRUDE, REVOLVE)

**prompt** A message on the command line or in a tooltip that asks for information or requests action such as specifying a point.

**proxy object** A substitute for a custom object when the ObjectARX application that created the custom object is not available. See also *custom object* and *object enabler*.

**QuickView** A tool to preview and switch between open drawings and layouts in a drawing.

**ray tracing** The renderer can generate reflections and refractions. Ray tracing traces the path of rays sampled from the light source. Reflections and refractions generated this way are physically accurate.

**ray-traced shadows** A way that the renderer can generate shadows. Ray tracing traces the path of rays sampled from the light source. Shadows appear where rays have been blocked by objects. Ray-traced shadows have sharp edges.

**rectangular break** To break a table into multiple parts that are evenly spaced and set at a user-specified height using the table breaking grips.

**redraw** To quickly refresh or clean up blip marks in the current viewport without updating the drawing's database. See also *regenerate*. (REDRAW)

**reference** A definition, known as an external reference or block reference, that is used and stored in the drawing. See also *block* (BLOCK) and *external reference* (xref). (XREF)

**refine** To quadruple the number of faces in a mesh object as you reset the baseline level of smoothness. (You cannot make a mesh courser than its baseline level.) You can also refine specified mesh faces without resetting the baseline level of smoothness for the object. (MESHREFINE)

**reflectance scale** Increases or decreases the amount of energy the material reflects.
reflection color  The color of a highlight on shiny material. Also called specular color.

reflection mapping  Creates the effect of a scene reflected on the surface of a shiny object.

refraction  How light distorts through an object.

regenerate  To update a drawing's screen display by recomputing the screen coordinates from the database. See also redraw. (REGEN)

region  Two-dimensional enclosed areas that have physical properties such as centroids or centers of mass. You can create regions from objects that form closed loops. They area commonly created in order to apply hatching and shading. (REGION)

relative coordinates  Coordinates specified in relation to previous coordinates.

relax constraints  Ability to temporarily ignore constraints while editing geometry. After the geometry is edited, the constraints are either removed or retained based on whether the constraint is still valid for the edited geometry.

RGB  For red, green, and blue. A system of defining colors by specifying percentages of red, green, and blue.

roll arrows  Curved arrows located above the ViewCube tool with which you can rotate the current view 90 degrees clockwise or counterclockwise.

roughness  Value to simulate how light hitting a face is reflected back to the user. A high roughness value simulates a non-shiny or rough object (sandpaper/carpet). A low roughness value simulates a very shiny object (metals, some plastics.)

row  A horizontally adjacent table cell selection spanning the width of the table. A single row is one cell in height.

rubber-band line  A line that stretches dynamically on the screen with the movement of the cursor. One endpoint of the line is attached to a point in your drawing, and the other is attached to the moving cursor.

running object snap  Setting an Object Snap mode so it continues for subsequent selections. See also Object Snap mode and object snap override. (OSNAP)

sampling  Sampling is an anti-aliasing technique. It provides a "best guess" color for each rendered pixel. The renderer first samples the scene color at locations within the pixel or along the pixel's edge, then uses a filter to combine the samples into a single pixel color.
scale representation The display of an annotative object based on the annotation scales that the object supports. For example, if an annotative object supports two annotations scales, it has two scale representations.

script file A set of commands executed sequentially with a single SCRIPT command. Script files are created outside the program using a text editor, saved in text format, and stored in an external file with the file extension .scr.

secondary table fragment Any fragment of a broken table that does not contain the beginning set of rows.

selection set One or more selected objects that a command can act upon at the same time.

shadow maps A shadow map is a bitmap that the renderer generates during a pre-rendering pass of the scene. Shadow maps don't show the color cast by transparent or translucent objects. On the other hand, shadow maps can have soft-edged shadows, which ray-traced shadows cannot.

Shadow mapped shadows provide softer edges and can require less calculation time than ray-traced shadows, but are less accurate. On the Advanced Render Settings palette, shadow mapped shadows are active when Shadow Map is turned on.

ShapeManager ShapeManager is the Autodesk technology that provides 3D solid modeling to AutoCAD and other products.

shortcut keys Keys and key combinations that start commands; for example, Ccommand-S saves a file. The function keys (Fn-F1, Fn-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.

sky The background color of the drawing area when perspective projection is turned on. The sky displays with a color gradient between the sky horizon (nearest to the horizon) and the sky zenith (opposite the horizon). See also ground plane.

smooth shading Smoothing of the edges between polygon faces.

smoothness A property of mesh objects that controls the roundness of the object. Objects with higher levels of smoothness have more faces, or tessellations.

snap angle The angle that the snap grid is rotated.
snap grid  The invisible grid that locks the pointer into alignment with the grid points according to the spacing set by Snap. Snap grid does not necessarily correspond to the visible grid, which is controlled separately by GRID. (SNAP)

Snap mode  A mode for locking a pointing device into alignment with an invisible rectangular grid. When Snap mode is on, the screen crosshairs and all input coordinates are snapped to the nearest point on the grid. The snap resolution defines the spacing of this grid. See also Object Snap mode. (SNAP)

snap resolution  The spacing between points of the snap grid.

solid history  A property of a solid that allows you to see and modify the original forms of the solid.

solid object  An object that represents the entire volume of an object, for example a box.

solid primitive  A basic solid form. Solid primitives include: box, wedge, cone, cylinder, sphere, torus, and pyramid.

spatial index  A list that organizes objects based on their location in space. A spatial index is used to locate what portion of the drawing is read when you partially open a drawing. Saving a spatial index with a drawing also enhances performance when working with external references. The INDEXCTL system variable controls whether layer and spatial indexes are saved with a drawing.

specular reflection  The light in a narrow cone where the angle of the incoming beam equals the angle of the reflected beam.

spline-fit  Uses the vertices of the selected polyline as the control points, or frame, of a curve approximating a B-spline. This curve, called a spline-fit polyline, passes through the first and last control points unless the original polyline was closed.

split face  A mesh face that has been subdivided along a specified vector.

STB file  For plot style table file. Contains plot styles and their characteristics.

sub-prompt  A command prompt that instructs for a form of input to complete a command or alter a property.

subdivision  A division, or tessellation in a mesh object. As a mesh object is smoothed, the number of subdivisions increases.

subobject  A part of a composite object.

surface  A surface is a 3D object that is a thin shell. Surfaces do not have mass or volume as 3D solids do. There are 3 types of surfaces: analytic, generic, and NURBS.
surface associativity  See associative surfaces

surface normal  Positive direction perpendicular to the surface of an object.

swept solid/surface   A solid or surface created in the shape of the specified profile (the swept object) swept along the specified path.  (SWEEP)

symbol  A representation of an item commonly used in drawings. Symbols are inserted in drawings as blocks.

symbol table  See definition table and block definition table.

system variable  A name that is recognized as a mode, size, or limit. Read-only system variables, such as DWGNAME, cannot be modified directly by the user.

table  A rectangular array of cells that contain annotation, primarily text but also blocks. In the AEC industry, tables are often referred to as “schedules” and contain information about the materials needed for the construction of the building being designed. In the manufacturing industry, they are often referred to as “BOM” (bills of materials).  (TABLE)

table break  The point at the bottom of a table row where the table will be split into a supplementary table fragment.

table style  A style that contains a specific table format and structure. A table style contains at least 3 cell styles.

temporary files  Data files created during an program session. The files are deleted by the time you end the session. If the session ends abnormally, such as during a power outage, temporary files might be left on the disk.

tessellation lines  Lines that help you visualize a curved surface.

In a 3D mesh object, tessellations indicate the boundaries of the mesh faces.

text style  A named, saved collection of settings that determines the appearance of text characters—for example, stretched, compressed, oblique, mirrored, or set in a vertical column.

texture map  The projection of an image (such as a tile pattern) onto an object (such as a chair).
thaw  A setting that displays previously frozen layers. See also freeze. (LAYER)

thickness  The distance certain objects are extruded to give them a 3D appearance. (PROPERTIES, CHPROP, ELEV, THICKNESS)

tiled viewports  See model viewports.

TILEMODE  A system variable that controls whether viewports can be created as movable, resizable objects (layout viewports), or as nonoverlapping display elements that appear side-by-side (model viewports). See also viewport.

tooltip  A small box of text that identifies or explains an object or interface element when the cursor hovers near or over it.

tracking  A way to locate a point relative to other points on the drawing.

translucency  How light is scattered through an object.

transmittance scale  Increases or decreases the amount of energy a transparent material transmits out to the scene.

transparency  A quantity defining how much light is let through an object.

transparent command  A command started while another is in progress. Precede transparent commands with an apostrophe.

two sided material  The positive and negative normal of the material will be considered during the rendering process.

UCS  See user coordinate system (UCS).

UCS icon  An icon that indicates the orientation of the UCS axes. (UCSICON)

underconstrained geometry  Objects with unsolved degrees of freedom are underconstrained.

underground  The XY plane of the user coordinate system when perspective projection is turned on and when viewed from below ground. The underground plane displays with a color gradient between the earth horizon (nearest to the horizon) and the earth azimuth (opposite the horizon). See also ground plane and sky.
**up direction** A vector defining what direction is Up. By default this is the positive $Z$–axis (0,0,+1).

The up direction and the north direction are always constrained such that they are perpendicular to each other.

**user coordinate system (UCS)** A user-defined coordinate system that defines the orientation of the $X$, $Y$, and $Z$ axes in 3D space. The UCS determines the default placement of geometry in a drawing. *See also* world coordinate system (WCS).

**user parameter** Named user-defined variable (real number or an expression) that can be used in expressions for dimensional constraints or other user parameters.

**UVW** The material’s coordinate space. Used instead of $XYZ$ because that is usually reserved for the world coordinate system (WCS). Most material maps are a 2D plane assigned to a 3D surface. The $U$, $V$, and $W$ coordinates parallel the relative directions of $X$, $Y$, and $Z$ coordinates. If you look at a 2D map image, $U$ is the equivalent of $X$, and represents the horizontal direction of the map. $V$ is the equivalent of $Y$, and represents the vertical direction of the map. $W$ is the equivalent of $Z$ and represents a direction perpendicular to the $UV$ plane of the map.

**vector** A mathematical object with precise direction and length but without specific location.

**vertex** A location where edges or polyline segments meet.

**view** A graphical representation of a model from a specific location (viewpoint) in space. *See also* viewpoint and viewport. (3DORBIT, VPOINT, DVIEW, VIEW)

**view category** A named collection of views in a sheet set that is often organized by function. *See also* subset.

**ViewCube** User interface element that displays the current orientation of a model, and allows you to interactively rotate the current view or restore a preset view.

**viewpoint** The location in 3D model space from which you are viewing a model. *See also* view and viewpoint. (3DORBIT, DVIEW, VPOINT)

**viewport** A bounded area that displays some portion of the model space of a drawing. The TILEMODE system variable determines the type of viewport created. 1. When TILEMODE is off (0), viewports are objects that can be moved and resized on a layout. (MVIEW) 2. When TILEMODE is on (1), the entire drawing area is divided into non-overlapping model viewports. *See also* TILEMODE, view, and viewpoint. (VPORTS)

Glossary | 1033
viewport configuration A named collection of model viewports that can be saved and restored. (VPORs)

virtual screen display The area in which the program can pan and zoom without regenerating the drawing.

visual style A collection of settings that control the display of edges and shading in a viewport.

volumetric shadows A photorealistically rendered volume of space cast by the shadow of an object.

watertight A closed 3D solid or mesh that has no gaps.

WCS See world coordinate system (WCS).

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 printing.

wireframe model The representation of an object using lines and curves to represent its boundaries.

working drawing A drawing for manufacturing or building purposes.

working set A group of objects selected for in-place reference editing.

workplane Another name for the XY plane of the user coordinate system. See also elevation and user coordinate system (UCS).

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).

X,Y,Z point filters See coordinate filters.

xref See external reference (xref).

zoom To reduce or increase the apparent magnification of the drawing area. (ZOOM)
Index

2D Cartesian coordinates 199
    coordinate filters 241
    entering 201
2D coordinates
    Cartesian 199
    entering 200
    polar 199
2D isometric views 951
2D objects
    flattened views of 3D objects 650
    sectioning 647
    simplified display 192
    simulating 3D 951
2D polar coordinates 199, 203
2D UCS icon 221
2D wireframe visual style 93
3D arrays 329
3D Cartesian coordinates
    coordinate filters 241
    defining 3D views 90
    entering 205
3D coordinates
    Cartesian coordinates 205
    cylindrical coordinates 207
    entering 205
    spherical coordinates 209
3D meshes
    about 511
    best converting objects to 534
    converting to 3D solids 632
    creases 620–621, 629
    creating 511
    creating from other objects 528
    custom meshes 537
    density 617
    editing 610
    extruding 622, 631
    faces 611
    facets 612
    gizmos 629
    grip editing 613
    legacy meshes 537
    merging faces 613
    mesh modeling 612
    predefined meshes 540
    primitives 515
    properties 592
    refining 619, 628
    repairing holes 613
    selection filters 630
    self-intersections 632
    smoothness 615
    spinning edges 613
    splitting faces 622, 628
    tessellation 512, 615
    types of 529
3D models
    3D solids 461
    3D surfaces 485
    3D views 107
    about 441, 547
    advantages 443
    backgrounds 99
    converting to meshes 547
    converting to objects 547
    cross sections 635
    dynamic views 110
    edge display 101
    editing 547
    flattened views of 650
    gizmos 561
    grips 561
    history 590
    interference problems in 484
    live sectioning 642
    meshes 610, 628
    navigating views 108
    parallel views 85
    performance issues and 105–106
    perspective views 85
    properties 590
    rendering 993
    shadows 99
subobjects 570
thickness 544
types of 441
viewing 548
visual styles 93
weblights 967
wireframes 542
3D Move gizmo 549, 554
3D objects
aligning 325
arrays 329
AutoCAD LT functionality 935
coordinates 205
display 101
exporting 926
extending 340
filleting 347
flattened views of 650
live sectioning 642
lofting 568
mirroring 336
modifying 547
rendering 993
rotating 324
shadows 100
simulating in 2D 951
smootheness 999
subobjects 570
surfaces 485
sweeping 567
thickness 544
trimming 340
visual styles 93
wireframes 542
3D Orbit tool 108
3D point clouds
AutoCAD LT functionality 937
3D printing 927
3D Rotate gizmo 549, 556
3D Scale gizmo 558
3D solid edges
imprinting objects on 589
modifying 570, 576
redundant edges 586
3D solid faces
imprinting objects on 589
modifying 570, 573
redundant faces 586
rendering 994
3D solid vertices
modifying 570, 580
redundant vertices 586
3D solids
calculating geometric data for 254
chamfering 577
cleaning 586
colors 575
composite solids 481, 581
converting meshes to 632
converting objects to 446, 477
converting surfaces to 480
converting to objects 547
coordinates 205
creating 461
dges 586
edges on 101, 570
exporting 927
extruding 450, 567
faces 570
filleting 577
flattened views of 650
gizmos 561
grips 561, 567
history 582, 590
imprinting objects on 589
interferences 484
lofting 456, 568
mass properties 256
meshes 528, 540
modifying 547
polysolids 475
pressing or pulling areas 587
primitives 590
properties 590
revolving 459, 568
separating into original shapes 585
shells 585
slicing 483
smootheness 999
solid primitives 462, 465
subobjects 570
surfaces 485
sweeping 453, 567
calculating 252
constraints 386
defining 3D views with 90
polar angles 239
polar coordinates 203
rotating objects by 323
snap angle 235
spotlights 982
unit types 200
angular constraints 387
angular dimensions
breaks in 822
creating 814
definition points 833
spacing between 825
angular units 56
anisotropic light distributions 967
annotation objects
annotative styles 662
blocks as 667
creating 660
defined 657
dimensions as 665, 801
hatches as 669
leaders as 666
model space and 131
multileaders as 666
orientation 673
previous release formats and 928
scale representations 671
scaling 132, 657–658
text 664, 706
tolerances as 665
types of 655
updating 659
updating scale 671
visibility 670
visual fidelity 68, 661, 928
annotational constraints 391, 394
annotative dimensions 801
Annotative property 657, 660
aperture box (AutoSnap) 229
append extend surfaces 598
applications
exporting files to 923
importing files from 907
ObjectARX 937
opening attribute extraction files 430
rendered images and 1005
arc length dimensions 818, 822
archiving
drawings with xrefs 892
arcs
dimensioning 809, 814, 818
drawing 270
fillet arcs 347
joining to polylines 357
linetypes 180
modifying 293
polyline arcs 276
PostScript rendering 925
rendering 999
revision cloud segments 292
tangent to lines 273
areas
calculating 253, 289
combining calculations 255
subtracting 255
arrays of objects
3D objects 329
creating 327
polar 328
rectangular 328
arrowheads
customizing 781
dimension arrowheads 769, 780
leader styles 728
artificial lighting 960
Asian fonts 742, 748
Asian set 742
association points of dimensions 836
associative dimensions
about 771
annotation objects and 665
association points 836
changing associativity 836
leader objects and 726
limitations 771
modifying 832, 835
previous release formats and 932
updating 771
associative hatches
  creating 680
  defined 676
  exploding 354
  extents 699
associative surfaces
  about 486
  creating 449, 505
associative text 724
atmospheric rendering effects 1001
attached xrefs
  demand loading and 891
  Internet-based xrefs 948
  relocated xrefs 889
  updating 890
attaching
  block attributes 420, 424
  files to drawings 907
  raster images 911
  xrefs 883, 886
attenuation of lights 984
attribute definitions 423
attribute extraction templates 426
attribute tags 423
attributes of blocks 420
auditing drawings
  examples 76
Auto-list feature 716
AutoCAD
  previous releases 928
  working in AutoCAD and AutoCAD LT 934
Autodesk Authorized Training Centers 4
Autodesk Consulting 5
Autodesk Developer Network (ADN) 5
Autodesk e-learning program 5
Autodesk Official Training Courseware 5
Autodesk products
  information resources 4
  late-breaking information about 5
Autodesk VIZ 1005
AutoFill feature 764
automatic saves 77
AutoSnap 226, 229
AutoStack feature 721
AutoTrack feature 244
axes
  coloring 43
  constraining movement to 554
  constraining rotation to 557
  constraining scale to 558
  positive direction of 214
  right-hand rule 214
  WCS 210
B
B-splines 502
back faces of 3D solids 994
back, moving objects to 196
background colors
  drawings 43
  hatches 685
  text 712
backgrounds
  rendering effects 1001
  visual styles and 99
backup files
  restoring drawings from 77
  saving 67
base points
  snap base point 235
baseline dimensions 806
basic dimensions 797
beam angle (spotlights) 982
beveled corners 348
Big Fonts
  properties 742
bills of materials (BOM) 757
binding
  dependent named objects to
drawings 900
  object definitions 900
  xrefs to drawings 892
bisecting objects with construction lines 288
bitmaps (BMP images)
  exporting 924
  importing 908
bitonal raster images 916
blending surfaces 495
blips 309
Block Attribute Manager

duplicate tags 435
reordering prompts 424

block attributes
about 420
annotative 660
attaching 424
attribute definitions 423
attribute extraction templates 426
attribute tags 423, 427, 435
attribute values 435
constant 421
definitions 434
editing 424
error handling 431
exporting data 426
extracting data from 425
invisible 421
modifying 433
nested blocks 431
removing 435
stand-alone 425
updating 435
variable 421

block definitions
about 413
annotations 655, 667
base points 413
block attributes 420
block libraries 419
colors 416
copying 420
creating 413
cross sections as 647
drawing files as 407, 414
fields in 735
inserting blocks 407
linetypes 416
lineweights 416
model space settings 132
modifying 431, 897
object properties 416
removing 411
updating 432

block libraries
creating 419

inserting blocks 408
updating block definitions 432

block references
annotations and 667
associative dimensions and 772
block attributes 420
clipping 894
colors 417
editing 353, 431
exploding 354, 437
grips 320
groups compared to 304
inserting 407
layers and 406
leader objects and 729
linetypes 417
model space settings 132
nested 418
orientation 673
paper space and 415
pasted text as 723
properties 416
scaling 407
table cells and 762
updating 435
xrefs vs. 415

blocks 413

BMP files (bitmaps)
exporting 924
importing 908

BOM (bills of materials) 757

borders (table cells) 761

boundaries
boundary edges 337
gaps in 290, 682
layout viewports 146
polylines 264, 277
recreating 701
regions 289
zooming to 82

boundary sets
defined 264
hatches 682
polylines 277
bounded areas 587
bounded hatches 676
boxes
  3D solids 465
  mesh boxes 516
  modifying 590
breaking
  objects 351
breaks in dimensions 822
brightness 915
bringing objects to front 196
bulge magnitude 488
bulleted lists 715
bullets 715
Buzzsaw location shortcuts 947
Buzzsaw project collaboration service 947
Buzzsaw sites 947
BYBLOCK setting
  block definitions 417
  properties 163
BYLAYER setting 163
C
calculating
  angles 252
  areas 253
  circumferences 254
  command prompt calculator 258
  distances 252
  model space scale factors 133
  perimeters 254
  point coordinates 252
calculators
  command prompt calculator 257
callout blocks
  annotations 655, 666
CALS files 908
cameras
  section objects and 646
canceling commands 308
candelas 972
caps 818
CDF files (comma delimited) 429
cells (tables)
  address format 764
  AutoFill feature 764
circular references between xrefs 899
circumferences 254
clamp curves 505
cleaning
  3D solids 586
  clearing screen 44
  screen display 309
clearing screen 44
Clipboard (Windows) 310
clinking
  blocks 894
  layout viewport boundaries 146
clipping boundaries
  blocks 894
  layout viewports 146
  raster images 913–914
  xrefs 884, 894
clipping planes
  3D views and 110
closed areas 587
closed curves 505
closed meshes 537
closed polylines 276
code pages 945
coincident grips 357
collaborative web sites 947
collected leader lines 726
color books 179
color libraries 176
color palettes 176
color-dependent plot style tables 858
  about 874
  plot styles in 873
color-dependent plot styles 873
colors
  3D solids 575
  ACI colors 176
  application window elements 43
  applying 176
  blocks 416–417
  changing 178
  color books 179
  color palettes 176
  current color 176–177
  defaults 168
  DIC color guide 177
  dimension elements 775
  display speed and 921
  edges 578
  filtering selection sets by 299
  layers and 165, 168, 171
  lighting 960, 963, 965, 982–983
  masking objects with 702
  names 176
  NURBS surfaces 608
  Pantone color books 177
  RAL color sets 177
raster images 909, 916
screening 151
shades 685
table elements 761
tints 685
True Colors 176
visual styles and 98
xref layer properties 884
columns (multiline text) 722
columns (tables)
  formatting 761
  modifying 757
combining
  regions 289
  solids 481, 581
comma-delimited files (CDF) 429
command line
  colors in 43
  switches 49
  syntax 50
command line switches
  customizing program startup 49
command prompt calculator 258
commands
  canceling 308
  command prompt calculator 258
  Dynamic Input and 223
  searching for 11, 13, 22
compass (ViewCube) 112
composite regions 289
composite solids
  3D solids 481, 581
  history 582
  modifying 584
  modifying original components 584
  selecting subobjects 562
  separating into original shapes 585
composite surfaces 584
composite tolerances 841
compound objects 353
compression
  JPEG files 924
Conceptual visual style 93
cones
  3D solids 468
  mesh cones 517
Index | 1043

modifying 590
configuring
  Render 1001
conflicting xref names 900
conical helixes 286
constant block attributes 421
Constrained Orbit tool 108
constraining
  object movement 554
  object rotation 557
  object scaling 558
constraint bars 380
constraint icons 379
constraint points 375
constraints
  annotational constraints 391
  applying 389
  applying automatically 384
  AutoCAD LT functionality 935
  constraint bars 380
  constraint points 375
  dimensional constraints 386
  displaying 379, 394
  dynamic constraints 390
  editing 377, 395
  editing constrained objects 381, 396
  fix constraints 376
  formulas 398
  fully constrained drawings 368
  geometric constraints 371, 373
  inferring 384
  multiple constraints 376
  object snaps and 384
  parameters 398–399
  parametric constraints 367
  reference parameters 391
  relaxing 369
  removing 369
  surfaces 508
  unconstrained drawings 368
  underconstrained drawings 368
construction lines
  drawing 287
  modifying 293
contextual fields 735
continued dimensions 806
continuity (surfaces) 488, 602, 606
contrast in raster images 915
control points on splines 282, 360, 504
control vertices 600–601
converting
  3D models to objects 547
  dimension units 793
  dimensions to annotations 665
  drawing units 54
  drawings to other formats 923
  files to AutoCAD LT 934
  fonts 745
  meshes to 3D solids 632
  nonassociative dimensions to associative 836
  objects to 3D solids 477
  objects to meshes 534
  objects to surfaces 501
  surfaces to 3D solids 480
  text to annotations 664
Coons surface patch meshes 532
coordinate filters 241
coordinate systems
  origin 199
  types of 199
  UCS (user coordinate system) 210
  WCS (world coordinate system) 210
coordinates
  2D coordinates 200
  3D coordinates 205
  absolute values 199
  calculating 252
  coordinate filters 241
  Dynamic Input and 223
  relative values 199
  types of 199
  unit types 200
coplanar faces 997
copying
  arrays of objects 327
  block definitions 420
  demand loading and 905
  edges 578
  faces 574
  formulas to table cells 764
  grip modes and 318
hatch properties 697
multiple copies of objects 327
object properties 165
objects 310, 326
objects to other drawings 67
offsetting objects 333
corners
beveled 348
chamfering 348
filleting 344
correcting
mistakes 307
spelling 754
CPolygon selection 296
creases
limiting distortion 629
mesh objects 620–621
cross sectioning
2D and 3D sections 647
live sectioning 642
lofting objects and section objects 456, 568
crosshairs
coloring 43
crossing selections 296
CTB files 858, 873
current viewports 124
cursor
Dynamic Input and locking movement 223
moving in tables 762
polar tracking 238
rubber-band line 237, 246
curved objects
analyzing 608
clamp curves 505
closed curves 505
drawing 270
modifying 293
open curves 505
rendering 999
custom objects 937
Customer Involvement Program 6
customizing
arrowheads 781
dimension text 790
hatches 685
object selection 300
visual styles 95
cutaway views 110
cutting
cutting edges 337
objects 310
slicing 483
cutting edges 337
cutting planes
live sectioning 642
section objects 635
slicing solids 483
cylinders
3D solids 469
mesh cylinders 519
modifying 590
cylindrical coordinates 207
cylindrical helixes 285

D
damaged drawing files 75
data extraction
block attributes and 425
data types in tables 761
databases
block attribute data in 426
datum
composite tolerances 841
geometric tolerances 840
ordinate dimensions 816
deleting
3D solid history 582
back faces 994
block attributes 435
block definitions 411
clipping boundaries 915
constraints 369
creases 622
dimension breaks 824
edges 577
faces 574
fillets 346
geometric constraints 373
group definitions 307
hidden surfaces 994
layer property overrides 174
layers 170, 172
linetypes 181
multiline vertices 365
objects 309-310
objects from groups 306
points in splines 360
text styles 739
unused named objects 309
vertices 581
demand loading (xrefs)
about 904
temporary xref file copies 905
updating xrefs 891
density of mesh objects 617
depth cueing 1001
deselecting objects 294, 297
DesignCenter
blocks in 419
copying block definitions 432
inserting blocks from 408
raster images in 911
detaching
raster images 913
xrefs 890
deviation tolerances 797
diameter constraints 387
diameter dimensions 809
creating 809
definition points 833
text 784
DIC color guide 177
dimension lines
angular dimensions 814
defined 769
leader objects vs. 724
linear dimensions 803
Dimension Style Manager
creating styles 773
dimension values 793
fractions 800
text options 783
text styles 790
dimension styles 773
about 773
applying 827
listing 774
modifying 775
overriding 828
system variables 774
dimension text 783
dimensional constraints
about 386
annotational constraints 391
applying 389
displaying 394
dynamic constraints 390
editing 395
editing constrained objects 396
illustrated 367
parameters 399
reference parameters 391
surfaces 508
dimensional input 224
dimensions 767
about 767
alignment 787, 825
angular 814
annotations 655, 665, 801
arc length 818
arrowheads 769
associative 771, 832, 835
basic 797
breaks in 822
creating 767, 803
customizing contents 790
dimension lines 769, 775
dimensional constraints 386
elements of 769
exploded 354, 771, 834
extension lines 769
grips 832
inspection 821
jog lines 820
lateral 797
linear 803
model space settings 132
modifying 353, 819
nonassociative 771, 833, 836
ordinate 816
previous release formats and 929, 932
radial 809
scale 801
spacing between 825
styles 773, 827
text 769, 783, 830
text styles 790
types 767, 803
units of measurement 790, 792
values 792
direct distance entry 200, 247
direction
axes 214
disassociated dimensions 833
discontinuities 601
display drivers 105
display performance 192, 921
displaying
3D solid history 582
alignment paths 245
annotative objects 670
constraints 379
dimensional constraints 394
draw order 196
drawing properties 60
drawing units 131
gizmos 549, 551
grid 233
grip tools (gizmos) 549
grips 314
isolated objects 197
layers 167
lights 978
linetypes 185
lineweights 188
originals in composite solids 582
overlapping objects 196
plot areas 850
Quick View images 64
raster images 921
rendered images 1004
shadows 100
simplifying 192
distances
calculating 252
equal intervals 248
fuzz distance 357
polar distances 239
specifying 247
distant lights 960, 977, 1002
distributing
dimensions 825
leader lines 726
dividing
objects into equal segments 249–251
objects into original shapes 585
docked palettes 48
doglegs 724, 809
donuts
drawing 279
exploding 354
simplified display 192
double click actions
editing 311
draft analysis 603, 609
draft quality display of raster images 921
draw order
changing 196
hatches 696
drawing
3D solids 465
arcs 270
boxes 465
circles 274
cones 468
construction lines 287
curved objects 270
cylinders 469
direct distance entry 247
donuts 279
eclipses 280
helixes and spirals 285
isometric circles 280, 953
layout viewports and 144
lines 261
multilines 267
object properties 161
objects 261
Ortho mode 237
parametric drawing 367
polygons 265, 276
polyline arcs 276
polylines 262
polysolids 475
precision 199
pyramids 472
rays 287
rectangles 265
reference points 286
revision clouds 291
spheres 471
splines 281
squares 266
tangent arcs and lines 273
tori 474
triangles 266
viewports and 124
wedges 466
wide polylines 276
drawing extents 82, 850, 861
drawing files (DWG)
    backups 77
    converting other files to 908
    file locations 73
    finding 60, 72
    inserting as blocks 407
    opening 61
    previous release formats 928
    repairing 75
    saving 67–68
drawing interchange format (DXF files)
    converting to DWG 908
    exporting 923
drawing orientation 849, 864
drawing properties
    displaying 60
    finding files with 60
drawing templates
    creating 58
    opening 58
    saving 68
    starting drawings with 58
drawing units
    about 54
    angular units 56
    converting 55
linear units 56
model space settings 131
precision 57
rounding off 57
drawings
    2D sections of 3D models 635
    3D views 107
    annotations in 655
    archiving 892
    background colors 43
    colors 176
    converting to AutoCAD LT 934
    copying between 67
    cross sectioning 635
    exporting 923
    extents 82
    file formats 68
    file locations 63, 73
    finding 60, 72
    fitting on paper 863, 866
    fully constrained drawings 368
    importing files into 907
    incremental saves 69
    information about 7
    inserting as blocks 414
    international files 69
    Internet-based 944
    layers 165
    layouts 137, 139
    lighting 955
    measurement units 54
    model space 129
    multiple open drawings 64
    naming 69
    object properties 161
    opening 61, 944
    orientation 849, 864
    page setups 845
    partial saves 68
    plot settings 845
    plotting 857
    previewing 64, 876
    previous release formats 931
    publishing 947
    rendering 993
    repairing damaged drawings 75
saving 67, 944
saving as templates 58
scaling 131
sharing 943
starting 53
switching between 65
templates 58
text in 706
unconstrained drawings 368
underconstrained drawings 368
viewports 121
views 81
xrefs 881
duplicating objects 326
DWG to PDF driver 877
DWT files
  saving layouts as 157
  starting drawings 58
DXB files
  converting to DWG 908
DXF files (drawing interchange format)
  converting to DWG 908
  exporting 923
  exporting block attribute data 429
  saving 68
DXX files 429
dynamic block definitions
  about 409
  AutoCAD LT functionality 936
  block attributes 420
  inserting blocks 407
dynamic block references
  about 409
  block attributes 420
dynamic constraints 390, 394
dynamic panning 81
dynamic prompts 225
dynamic UCSs 218
dynamic viewing 110

E
e-learning program (Autodesk) 5
earlier versions of AutoCAD 931
edge-defined meshes 532
edges
  3D solid edges 570
  boundary edges 337
  colors 578
  copying 578
  creases 621
  customizing 101
  cutting edges 337
  deleting 577
  imprinting objects on 589
  mesh edges 532
  modifying 576
  redundant edges 586
  rendering 998
  surfaces 589
  trimming to 596
editing
  3D models 547
  3D solids 547, 573, 580
  block attribute definitions 424
  block attributes 433–434
  block definitions 431, 897
  block references 431
  columns of text 722
  complex objects 353
  composite solids 584
  constrained objects 381, 396
  constraints 377, 395
  custom objects 938
  dimension styles 775
dimensions 819
  edges 576
  faces 573
  fields in text 733
  fills 697
gizmos 549
grip modes 313
grip tools (gizmos) 549
groups 306
hatches 697
helixes 363
layer objects 168
layers 171
leader objects 725
meshes 610
multilines 364
objects 293, 311
polylines 355
revision clouds 292
separating solids into shapes 585
spirals 363
spline 360
surfaces 547, 570, 573, 584, 594
tables 757, 762
text 749
text styles 738
vertices 580
visual styles 94
xrefs 897
education products 884
ellipses
calculating geometric data for 254
drawing 280
grips on 315
modifying 293
PostScript rendering 925
elliptical arcs
modifying 293
PostScript rendering 925
EPS files
exporting 925
equations
parametric constraints 398
erasing objects 309
errors
block attribute extraction files 431
correcting 307
xref error messages 898
Excel spreadsheets
opening attribute extraction files 430
exploded dimensions 771, 834
exploding
blocks 437
compound objects 353
objects 353
exporting
3D solids 926–927
block attribute data 426
drawings 923
export file formats 923
plot files 877
rendered images 1004
expressions 258
Parameters Manager 400
parametric constraints 398
extending
3D objects 340
chamfered objects 349
filed objects 345
objects 337
extension lines
angular dimensions 814
appearance 777
arc length dimensions 818
arcs and 809
breaks in 822
defined 769
oblique 808
origins of 803
properties 777
text on 783, 787
external references (xrefs)
about 881
archiving drawings with 892
attaching 883, 886
Autodesk Educational Products 884
binding 892
blocks and 408, 415
circular references 899
clipping 894
defined 881
demand loading 891, 905
detaching 890
editing 897
error messages 898
exploding 354
fields in 735
Internet-based xrefs 948
layer properties 884
log files 901
missing 62, 898
name conflicts 900
nested 886
notifications about 889
overlays 886
paths 887–888, 905
reloading 890
relocated xrefs 889
temporary file copies 905
unloading 905
updating 881, 890
External References palette
  loading or unloading images 921
  raster image information 917
externally referenced dimension styles 774
extracting
  block attribute data 425–426
  geometric data 252
extruding
  faces 631
  mesh faces 622
  objects 567, 587
extruding objects 450

F

  face counts 998
  face normals 994
  face views 116
  faces of 3D solids
    colors 575
    coplanar faces 997
    copying 574
    creases 621
    edge display 101
    extruding 622, 631
    face styles 95
    imprinting objects on 589
    intersecting faces 996
    mesh faces 619, 998
    mesh objects 611
    modifying 570, 573
    redundant faces 586
    rendering 994
    splitting 622, 628
    facet edges 101
    faceted lighting 96
    faceted mesh objects 612
    faceted surfaces 528
    fade settings
      raster images 915
  falloff angle of spotlights 982
  feature classifications 938
  field angle of spotlights 982
  fields (databases)
    attribute extraction and 427
  fields (text)
    about 733
    AutoCAD LT functionality 934
    hyperlinks in 737
    previous releases of AutoCAD and 736
  file formats
    attribute extraction files 429
    export file formats 923
    importable files 907
    pasted objects 310
    plot files 877
    plottable formats 877
    PostScript formats 925
    raster formats 909, 924
    rendered images 1004
    saving drawings 67
    saving in older formats 931
  file input and output commands 944
  files
    backups 77
    block attribute data 426
    Buzzsaw sites and 947
    default locations 63
    exporting 923
    finding 60
    importing 907
    large objects in 63
    opening 61
    publishing 947
    repairing damaged files 75
    saving 67
    search paths 73
  Fill mode 192
  filleting
    3D solids 577
    objects 344
    surfaces 599
  fills
    boundaries 680, 701
    gradient fills 685
modifying 697
PostScript rendering 925
properties 693
solid fills 676, 685
turning off display 192
wipeout objects 702
filter colors of lighting 982
filtering
layers 170
subobject selection filters 565, 630
finding
commands 11, 13, 22
drawing files 60, 72
raster images 919
text 753
fit points in splines 360
fit points on splines 283, 504
fitting
dimension text 783
Fix constraints 376
fixed-length extension lines 778
fixtures (lighting) 956, 987
flat lighting 96
flatshot objects 650
flattened views of 3D objects 650
FLIC files 908
flipping
arrowheads 780
lights 979
objects 335
floating palettes 48
floating properties 416
FMP files (font mapping) 745
fog effects 1001
folders
file locations 63
font mapping files 744
fonts 740
alternative fonts 744
Asian set 742
assigning 740
Big Fonts 742
converting 745
file locations 73
international sets 742
mapping 745
PostScript 744
previous release file formats and 930
proxy fonts 741, 744
substitute fonts 744
TrueType 741
types of 740
Unicode 742
forcing internal lines 784
foreshortened radius dimensions 811
form tolerances 837
formatting text
character formatting 739
fields 733
multiline text 711, 714
stacked text 720
table cells 761–762
formulas
copying 764
inserting in table cells 764
parametric constraints 398
fractions 711, 720, 800
Free Orbit tool 108
free weblights 968
freezing
layers 167
layers in layout viewports 150
front views 90
front, moving objects to 196
FTP sites
opening Internet drawings 944
saving drawings to 944
full xref paths 888
fully constrained drawings 368
functions 401
fuzz distance 357

G
G0 and G1 continuity 489, 606
G2 curvature 489, 607
gaps between objects 352
gaps in boundaries 682
Gaussian curvature 608
Geo SPOT files 908
generic geometric constraints about 371
applying 373
applying automatically 384
constraint points 375
displaying 379
editing 377
editing constrained objects 381
fix constraints 376
illustrated 367
inferring 384
multiple constraints 376
relaxing 369
removing 369, 373
surfaces 508
geometric tolerances 837
composite tolerances 841
datum reference frames 840
lateral tolerances 797
material conditions 839
projected tolerances 840
geometry
object geometric data 252
projecting 597
rendering 996
trimming edges with 597
gizmos
3D meshes 613
3D models 561
about 549
displaying 551
mesh objects and 629
moving objects with 554
rotating objects with 556
scaling objects with 558
settings 552
switching 552
global linetype scale factor 184
goniometers 972
goniometric diagrams 968
Gooch face style 95
gradient fills
creating 685
simplified display 192
graphics cards 105
grid
displaying 232
snapping to 232
grid limits
calculating 133
overriding 233
zooming to 82
grip modes 313
grip tools (gizmos) 549
grips
3D meshes 613
3D solids 561, 567
blocks 320
coincident 357
constrained objects 382, 396
copying objects with 318
dimension text 830
dimensional input 224
dimensions 832
display options 314
editing objects with 313
gizmos 549
grip tools (gizmos) 549
hatches 699
helixes 363
jog lines 820
multi-functional 316, 355
non-associative hatch objects 316
polylines 316
quadrant grips 315
section objects 644
selecting 314
splines 316, 360
surfaces 561, 567
tables 758
text objects 750, 752
types of 562
groups
about 304
adding objects to 306
creating 304–305
editing groups 306
editing objects 306
removing group definition 307
removing objects from 306
reordering objects in 306
selecting 305
guide curves 448, 568
gutters between text columns 722
handdrawn effects 102
hanging indents 718
hardware
  acceleration 105
  linetypes and 180
hatches
  alignment 698
  annotations and 655, 669
  associative 676, 680
  background colors 685
  boundaries 680, 701
  bounded hatches 676
  chamfering boundaries 348
  customizing 685
  density 691
  draw order 696
  editing 353
  exploding 354
  file locations 73
  filleting boundaries 345
  gradient fills 685
  islands within 680–681
  ISO patterns 685
  large or complex drawings and 682
  model space settings 132
  modifying 697
  nonassociative 681
  orientation 673
  origin point 689
  pattern libraries 685
  pre-defined 685
  properties 693, 697
  rotating 698
  scaling patterns 691, 698
  simplified display 192
  solid fills 676
  styles 680–681
  transparency 693
  unbounded hatches 675, 694
hats 818
height
  table rows 757
  text 715
helixes
  drawing 285
  modifying 363
Help
  late-breaking product information 5
  Readme topic 5
hexagons 266
hidden objects 854, 868
hidden surfaces 994
Hidden visual style 93
hiding
  annotative objects 670
  hatch boundaries 694
  image boundaries 913
  isolating objects 197
  layers 167
  paper space objects 870
  raster images 921
highlighting
  objects 301
  suppressing 922
  visual styles and 97
histories of 3D solids 582, 590
holes in 3D solids 587
holes in meshes 613
hollow objects 585
Home view 118
hook lines 724, 809
horizontal dimension text 787
horizontal dimensions 804
horizontal text 748
hotspot angle of spotlights 982
hyperlinks
  fields and 737

I

IES photometric data files 968, 971
IG4 files 908
imperial units
  converting to metric 54
  linetype definition file for 181
importing
  files into drawings 907
  named page setups 846
  text 723
lamp intensity 963, 965, 983
landings
dimension text 809
multileader leader lines 724, 726
landscape orientation 849, 864
languages
previous release file formats 930
Unicode text 945
working internationally 945
large objects 63, 70
lateral tolerances
displaying 797
stacked text in 720
layer properties
blocks and 417
xref properties 884
Layer Properties Manager
layer names 170
property overrides 173
layers
about 165
block elements and 406
colors 168, 171, 178
creating 169
current 170
default properties 168
deleting 170, 172
filtering 170
freezing 150, 167
hiding 167
layer 0 166
layout viewports and 150
linetypes 168, 171, 183, 884
lineweights 168, 171, 186, 191
live sectioning and 643
locking 167
modifying 171
modifying objects on 168
moving objects to another 171, 178, 183, 191
naming 169, 171
overriding properties 168, 173
plot styles 168, 876
preventing selection 298
previous release formats and properties 171
referred 170
selecting 170
settings 171
thawing 167
transparency 167, 194
turning on or off 167
undoing changes 171
unlocking 167
unused 170
visibility 144, 167
xrefs and 884
layout templates 157
layout viewports
about 137
accessing model space from 144
aligning views in 154
annotations and 657–658
creating 145
drawing in 144
freezing layers in 150
linetypes in 153
magnifying 144
modifying views in 144
multiple 145
plotting 854
resizing 147
rotating views in 155
scale 145
screening objects in 151
shaded 854
turning on or off 152
views in 147
visibility in 149
layouts
adding 142
dimensioning in 801
initializing 846, 858
layer visibility 144, 167
layout tabs 137
layout templates 157
layout viewports 137
lineweights in 188, 853
opening 64
orientation 673, 849, 864
page setups 845
paper size 848
plot scale 852
plot settings 845, 861, 865
plot style tables 874
plotting 857
previewing 64
shaded viewports 867
switching to model space 66
work process 142
leaders (leader lines)
   annotations 655, 666
   associativity and 724, 726
   automatically creating 784
   blocks in 729
   breaks in 822
   creating 725
   defined 724
   landings 726
   modifying 725
   multileaders 725
   multiline text in 729
   ordinate dimensions and 817
   styles 728
leading (line spacing) 719
least material condition values 839
legacy mesh types 534, 537
lengthening objects 341
lettered lists 715
LID (luminous intensity
distribution) 972
light glyphs
displaying 955
plotting 982
settings 978
lighting
   about 955
   assigning shapes to 977
   colors 960, 963, 965, 982–983
   default 955
   display options 978
   distant lights 977, 984
   distribution 970, 984
   face styles and 96
   fixtures 956
   guidelines 960
   highlights 97
   illuminating scenes 960
lamp values 975
light glyphs 955, 978
light intensity 967
luminaire objects 987
luminous intensity distribution
   (LID) 972
photometric lighting 956, 959
placement and location 979
point lights 962
properties 982
spotlights 965
Standard lighting 955, 959
visual styles and 94
weblights 967
workflow 959
limits
   arrays 329
   lighting 985
tolerances 798
LIN files (linetype library) 181
line spacing
   dimension text 830
   previous release formats and 929
   text 712, 719
linear constraints 387
linear dimensions
   breaks in 822
   creating 803
   definition points 833
   jog lines 820
   spacing between 825
   types of 803
linear units 56
lines
   dimension lines 775
   direct distance entry 247
drawing 261
   extension lines 777
   filleting 346
   freehand lines 269
   joining to polylines 357
   linear constraints 387
   linetypes 180
   linetypes on short segments 185
   lineweights 186
   modifying 293
<table>
<thead>
<tr>
<th>Term</th>
<th>Page(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>multilines</td>
<td>267</td>
</tr>
<tr>
<td>occluded</td>
<td>102</td>
</tr>
<tr>
<td>Ortho mode</td>
<td>237</td>
</tr>
<tr>
<td>polyline arcs</td>
<td>276</td>
</tr>
<tr>
<td>table gridlines</td>
<td>761</td>
</tr>
<tr>
<td>tangent to arcs</td>
<td>273</td>
</tr>
<tr>
<td>Linetype Manager</td>
<td>184</td>
</tr>
<tr>
<td>scale settings</td>
<td></td>
</tr>
<tr>
<td>linetypes</td>
<td></td>
</tr>
<tr>
<td>about</td>
<td>180</td>
</tr>
<tr>
<td>applying</td>
<td>182</td>
</tr>
<tr>
<td>blocks</td>
<td>416–417</td>
</tr>
<tr>
<td>current</td>
<td>182</td>
</tr>
<tr>
<td>defaults</td>
<td>168</td>
</tr>
<tr>
<td>deleting</td>
<td>181</td>
</tr>
<tr>
<td>displaying on short segments</td>
<td>185</td>
</tr>
<tr>
<td>file locations</td>
<td>73</td>
</tr>
<tr>
<td>filtering selection sets by</td>
<td>299</td>
</tr>
<tr>
<td>freehand sketches</td>
<td>269</td>
</tr>
<tr>
<td>hardware linetypes</td>
<td>180</td>
</tr>
<tr>
<td>layers and</td>
<td>165, 171, 183</td>
</tr>
<tr>
<td>layout viewports and</td>
<td>153</td>
</tr>
<tr>
<td>loading</td>
<td>181</td>
</tr>
<tr>
<td>model space settings</td>
<td>132</td>
</tr>
<tr>
<td>polylines</td>
<td>185</td>
</tr>
<tr>
<td>reapplying</td>
<td>183</td>
</tr>
<tr>
<td>scaling</td>
<td>153, 184</td>
</tr>
<tr>
<td>lineweights</td>
<td></td>
</tr>
<tr>
<td>about</td>
<td>186</td>
</tr>
<tr>
<td>applying</td>
<td>186</td>
</tr>
<tr>
<td>block properties</td>
<td>416</td>
</tr>
<tr>
<td>current</td>
<td>189</td>
</tr>
<tr>
<td>defaults</td>
<td>168</td>
</tr>
<tr>
<td>dimension elements</td>
<td>775</td>
</tr>
<tr>
<td>displaying</td>
<td>188</td>
</tr>
<tr>
<td>filtering selection sets by</td>
<td>299</td>
</tr>
<tr>
<td>layers and</td>
<td>165, 171, 191</td>
</tr>
<tr>
<td>model space and</td>
<td>186</td>
</tr>
<tr>
<td>objects not displaying</td>
<td>186</td>
</tr>
<tr>
<td>overriding</td>
<td>191</td>
</tr>
<tr>
<td>plotting</td>
<td>870</td>
</tr>
<tr>
<td>previous releases and</td>
<td>190</td>
</tr>
<tr>
<td>reassigning</td>
<td>191</td>
</tr>
<tr>
<td>regenerating drawings and</td>
<td>188</td>
</tr>
<tr>
<td>scale</td>
<td>187, 853</td>
</tr>
<tr>
<td>turning on or off</td>
<td>188, 193</td>
</tr>
<tr>
<td>listing</td>
<td></td>
</tr>
<tr>
<td>dimension styles</td>
<td>774</td>
</tr>
<tr>
<td>dimension system variables</td>
<td>774</td>
</tr>
<tr>
<td>raster images</td>
<td>917</td>
</tr>
<tr>
<td>xrefs</td>
<td>901</td>
</tr>
<tr>
<td>lists in multiline text</td>
<td>715</td>
</tr>
<tr>
<td>live sectioning</td>
<td></td>
</tr>
<tr>
<td>about</td>
<td>635, 642</td>
</tr>
<tr>
<td>cameras and</td>
<td>646</td>
</tr>
<tr>
<td>LMC symbol (least material condition)</td>
<td>839</td>
</tr>
<tr>
<td>location</td>
<td></td>
</tr>
<tr>
<td>default drawing folder</td>
<td>63</td>
</tr>
<tr>
<td>locking</td>
<td></td>
</tr>
<tr>
<td>cursor movements</td>
<td>237, 241</td>
</tr>
<tr>
<td>layers</td>
<td>167</td>
</tr>
<tr>
<td>layout viewport scale</td>
<td>145, 148</td>
</tr>
<tr>
<td>lofting objects</td>
<td></td>
</tr>
<tr>
<td>3D objects</td>
<td>456, 568</td>
</tr>
<tr>
<td>surfaces</td>
<td>568</td>
</tr>
<tr>
<td>log files</td>
<td></td>
</tr>
<tr>
<td>xrefs</td>
<td>901</td>
</tr>
<tr>
<td>logo startup screen</td>
<td>50</td>
</tr>
<tr>
<td>lumens</td>
<td>972</td>
</tr>
<tr>
<td>luminaire objects</td>
<td>956, 972, 987</td>
</tr>
<tr>
<td>luminous intensity distribution</td>
<td></td>
</tr>
<tr>
<td>(LID)</td>
<td>972</td>
</tr>
<tr>
<td>LZW compression</td>
<td>911</td>
</tr>
<tr>
<td>M</td>
<td></td>
</tr>
<tr>
<td>M symbol (maximum material condition)</td>
<td>839</td>
</tr>
<tr>
<td>magnifying views</td>
<td>81</td>
</tr>
<tr>
<td>major grid lines</td>
<td>233</td>
</tr>
<tr>
<td>manufacturer data</td>
<td></td>
</tr>
<tr>
<td>IES lighting data files</td>
<td>968, 971</td>
</tr>
<tr>
<td>lighting fixtures</td>
<td>956</td>
</tr>
<tr>
<td>mapping</td>
<td></td>
</tr>
<tr>
<td>fonts</td>
<td>745</td>
</tr>
<tr>
<td>markers</td>
<td></td>
</tr>
<tr>
<td>equal segments on objects</td>
<td>249–251</td>
</tr>
</tbody>
</table>
object snaps 229
point markers 250–251
tracking points 244
markups
revision clouds 291
masking objects 702
mass properties 256
master drawings 881
material condition symbols 839
MaxArray system registry variable 329
maximum material condition values 839
measurement units
about 54
coordinates 200
dimension text 790
dimensions 792
model space 131
plot scale 866
measurements
dimensions 767
equal intervals 249–251
tolerances 837
memory (RAM)
allocating 105
memory tuning 106
merge extend surfaces 598
merging
cells in tables 759
mesh faces 613
mesh models 443
meshes
about 511, 528
best practices 628
boxes 516
cones 517
converting objects to 534, 547
converting to 3D solids 478, 548, 632
creases 620–621, 629
creating 512, 528
custom 537
cylinders 519
density 617
editing 610
extruding 631
extruding faces 622
faces 611
facets 612
gizmos 629
grip editing 613
legacy mesh types 534, 537
merging faces 613
mesh modeling 612
predefined meshes 540
primitives 515
properties 592
pyramids 521
refining 619, 628
rendering 998
repairing holes 613
selection filters 630
self-intersections 632
smoothness 615
spheres 523
splitting faces 622, 628
tessellation 512, 615
tori 527
types 529
wedges 525
metric units
converting to imperial 54
linetype definition file for 181
Microsoft Excel spreadsheets
opening attribute extraction files 430
Microsoft Windows Clipboard 310
minor grid lines 233
mirroring
grip methods for 314
lights 979
objects 313, 335
text 335, 749
misspellings 754
MMC symbol (maximum material condition) 839
model space
accessing from layout viewports 144
annotations and 131, 657–658
defined 137
dimensioning in 801
drawing process in 129, 141
layer visibility settings 144
lineweight display in 186, 188
live sectioning and 642
moving objects to paper space 142
plot scale 865
plotting from 132, 862, 870
switching to layouts 66
viewports 121
model space viewports
assigning UCSs to 219
creating 121
UCSs in 216
modifying
3D models 547
3D solids 547, 573, 580
block attribute definitions 424
block attributes 433–434
block definitions 431, 897
block references 431, 897
colors 178
columns of text 722
complex objects 353
composite solids 584
constrained objects 381, 396
constraints 395
dimension styles 775
dimensions 819
edges 576
faces 573
fields in text 733
fills 697
gizmos 549
grip tools (gizmos) 549
hatches 697
helixes and spirals 363
layer objects 168
layers 171
layout viewports 145, 147
leader objects 725
lineweights 191
meshes 610
multilinees 364
objects 293, 311
polylines 355
revision clouds 292
separating objects into shapes 585
splines 360
surfaces 547, 570, 573, 584, 594
tables 757, 762
text 749
text styles 738
vertices 580
visual styles 94
xrefs 897
molds 609
moments of inertia 256
moonlight 960
moving
3D Move gizmo 549, 554
3D subobjects 570
constraining movement 554
dimension text 830
edges 576
faces 573
grip methods for 314
jog lines 820
layer objects to other layers 171
leader objects 726
lights 979
objects 313, 321, 333
objects between model and paper space 142
referenced drawings 889
stretch-moves 321
text 750–751
UCSs 210
vertices 580
mtext (multiline text) 711
multi-functional grips 316, 355
multilineers (leader lines)
annotations 655, 666
blocks in 729
breaks in 822
creating 725
landings 726
modifying 725
previous release file formats and 929
styles 728
text in 729
multiline text
about 711
aligning 713
annotations 655
columns 722
creating 711
editing 751
finding 753
formatting 714
height 715, 746
indenting 718
justification 713
leader objects and 724
line spacing 719
lists in 715
obliquing angle 747
orientation 748
previous release file formats and 929
properties 712
replacing 753
stacked 720
styles 714, 738
tabs 718
text wrap 712
multilines
drawing 267
editing commands 365
intersections 365
modifying 364
styles 267
vertices 365
multiple drawings
opening 64
switching between 67
multiple hot grip selection 314
multiple-view drawing layouts 137
My Documents folder 63, 73

dnamed views
saving 84
section objects and 646
naming
layers 169, 171
lights 982
raster images 919
selection filters 299
text styles 738
viewports 125
views 84
naming conventions
international 69
natural lighting 959
navigating
3D views 108
ViewCube 111
nested blocks
block attributes 431
clipped 895
creating 418
nested xrefs
clipped 895
defined 886
paths 887, 898
network surfaces 492–493
Non-Uniform Rational B-Splines surfaces (NURBS) 502
nonassociative dimensions 771, 833, 836
nonassociative hatches 681, 700
nonrectangular layout viewports 146
non-system plotter drivers
PDF output 877
nonuniform rational B-spline curves 281
normals of 3D faces
about 994
Not Found raster images 918
notes
annotations 655, 664
creating text 706
notification
relocated xrefs 889
numbered lists 715
numeric values
block attribute data 429

N

named objects
xref name conflicts 900
named page setups
applying 845
creating 846
named plot style tables
about 859, 873
plot styles in 876
named plot styles
about 873, 876
dimensions 792
rounding 795
suppressing zeros 796
NURBS curves 281, 601, 603
NURBS surfaces
about 486
analyzing 608
creating 442, 502
editing 594, 600
rebuilding 601

O
object enablers 938
object properties
about 161
assigning 161
blocks 416
colors 176, 178
copying 165
layer defaults 168
linetypes 180
lineweights 187
object snap tracking 244
object snaps
3D space 228
about 226
angles of 235
AutoSnap 229
base points 235
constraints and 384
grid snap 232
isometric drawings 951
object snap tracking 244
overriding 230
PolarSnap feature 238
running object snaps 227
shortcut menu 227, 229
ObjectARX applications
custom and proxy objects 937
objects
3D objects 441
aligning 324
arrays of 327
assigning to other layers 178, 183, 191

breaking 351
chamfering 348
colors 176
converting to meshes 534
converting to revision clouds 291
copying 310, 318, 326
creating 3D solids from 477
creating meshes from 528
creating solids or surfaces from 446, 501
cross sectioning 635
custom objects 937
cutting 310
deleting 310
deselecting 294, 297
draw order 196
drawing 261
editing 293, 311
equal intervals on 248
erasing 309
feature classifications 938
filleting 344
geometric data 252
grouping 304
highlighting 301
isolating 197
large objects 63, 70
layers 165
layout viewports and 144
linetypes 180
lineweights 186
magnifying 82
masking 702
mass properties 256
materials 989
mirroring 335
moving 321
offsetting 318, 333
overlapping 196
paper space and 142
pasting 310
plotting 870
proxy objects 937
reshaping 341
resizing 341
rotating 323
saving 68
scaling 342
section objects 635
selecting 293
separating into original shapes 585
simplified display 192
snapping 226
stretching 342
subobjects 570
textures 989
transparency 194
visibility 149
oblique extension lines 808
oblique text 747
obscured edges 101
occluded lines 102
offset snaps 318
offsetting
construction lines 288
copying objects 318
objects 333
plot offsets 851, 864
surfaces 498
temporary reference points 248
opacity
transparency settings 194
visual styles 97
open curves 505
open meshes 537
opening
attribute extraction files 430
drawings 61, 944
files in other formats 907
files with large objects 63, 70
layouts 64
multiple drawings 64
older files 928
operators
Parameters Manager 400
ordinate dimensions 816, 822, 833
orientation
annotations 673
drawings 849, 864
text 748
origin
coordinate systems 199, 210
displaying UCS icon at 221
hatches 689
ordinate dimensions 816
Ortho mode 237
orthographic projection view 117
orthographic sectioning 639
orthographic views 89
overhang edge effect 102
overlapping objects
3D solids 484
draw order 196
rendering 997
selecting 293, 564
overlying xrefs 886
overlays 165
overrides
angle overrides 241
center locations of dimensions 811
dimension styles 774, 828
grid spacing 235
layer property overrides 168, 173
linetypes 183
lineweights 191
object snaps 230
previous release formats and 929
removing 174
snap spacing 235
overshoot of extension lines 777

P

Page Setup Manager
creating page setups 845
page setups 845
named page setups 845–846
Page Setup Manager 845
plot settings 860
settings 845
pages
page setups 845
palettes
dock 48
float 48
icons 48
resize 48
Pan tool 109
Index | 1063

panning
  3D views 109
  about 81
  dynamic 81
  layout viewports 144
Pantone color books 177
paper size
  fitting drawings on 866
  layout settings 848
  plot scale and 865
  scaling drawings to fit 852
  settings 862
paper space 137
  about 137
  blocks and 415
  dimensioning in 801
  drawing in 137
  hiding objects 870
  layout work process 142
  lineweight display in 188
  moving objects to model space 142
  object visibility 149
  plotting options 870
  UCS limitations 217
paper-saving features 863
paragraphs (multiline text) 711
parallel dimensions 825
parallel lines
  construction lines 288
  filleting 347
parallel projections
  about 85
  creating 88
parameters
  constraints 399
  parametric constraints 398
parametric constraints
  about 367
  annotational constraints 391
  applying 389
  dimensional constraints 386
  displaying 379, 394
  dynamic constraints 390
  editing constrained objects 381, 396
  formulas 398
  geometric constraints 371, 373
parameters 398
  reference parameters 391
  relaxing 369
  removing 369
parametric drawing
  about 367
  parametric constraints 367
partial saves 68
Partner Products and Services
  (Autodesk) 5
parts
  inspection dimensions 821
  parts lists 426
pasting
  lists 717
  objects 310
  objects from other drawings 67
  text 723
patching surfaces 496
paths
  default file locations 63
  extruding 567
  lofting 568
  raster image files 919
  support files 73
  sweeping 453
patterns
  hatches 685, 691
  libraries 685
PCX files
  attaching 908
PDF files
  exporting 923
  plotting 877
performance improvement
  3D display and 105
  arrays and 329
  fills display 192
  groups and 305
  hardware acceleration 105
  incremental saves 69
  layers and 167
  lineweights and 188, 193
  memory and 105–106
  performance tuning 105
  raster image display 920
resolution  999
  shadow display  100, 982
  software acceleration  105
  text display  192
perimeters  254
perpendicular lines  237
perspective views
  about  85
  AutoCAD LT functionality and defining  937
photometric lights
  distant lights and  977
  goniometric diagrams  968
  IES data files  968, 971
  lamp values  975
  luminous intensity distribution
    (LID)  972
  overview  959
  point lights  963
  properties  963, 983
  spotlights  965
  weblights  967, 984
  workflow  956
photorealistic rendering  993
pickbox cursor  293
PICT files  908
pixels
  raster images  908
  removing strays on display  309
plan views
  changing viewpoints  91
  defined  90
planar closed spline curves  254
planar surfaces  492
planes
  constraining movement to  555
  constraining scale to  559
  workplanes  199
plot areas
  setting  850, 861
plot settings
  objects  870
  orientation  864
  page setups and  860
  scale  865
  transparency  870
  plot stamps
    about  859
    turning on  870
plot style tables
  assigning to layouts  853, 874
  predefined  875
  shaded viewport plotting options and  867
  types  858, 873–875
plot styles
  about  871
  layers and  168, 171
  object plot styles  876
  plot style tables  853, 873
  selecting  870
  types  858, 872
Plotter Configuration Editor
  paper size  848
plotters
  offsetting plots  864
  paper size  862
  paper-saving features  863
  selecting  848, 861
plotting
  draw order and  196
exporting files  877
  file formats  877
  fit options  852
  hardware linetypes  180
  layout process and  139
  light glyphs  982
  lineweights  187–188, 870
  model space settings  132
  offsets  851
  orientation  849, 864
  page setups  845, 860
  paper size  862
  paper space objects  870
  plot scale  865
  plot style tables  853
  plot styles  870–871
  positioning image on paper  863
  previewing  876
  resolution  869
  scaling drawings for  852

1064 | Index
screened objects 151
section objects 650
selecting output devices 861
test plot performance 192
text frames 193
transparent objects 195, 870
wipeout objects 703
PNG files (Portable Network Graphics)
exporting 924
importing 908
point clouds
AutoCAD LT functionality 937
point lights
artificial lighting 960
attenuation rates 984
location 980
overview 962
target points 962
point markers 250–251
pointer input 223
points
block base points 413
control points 360
coordinates 252
direct distance entry 247
drawing 286
equal intervals between 249–251
fit points 360
geometric data 252
offsetting objects from 248
point markers 250–251
specifying 246
tracking points 244
polar angles 239
polar arrays 328
polar coordinates
2D coordinates 203
about 199
cylindrical coordinates 207
spherical coordinates 209
polar tracking 238, 244
PolarSnap feature 238
polyface meshes
about 537
creating 539
rendering 998
polygon meshes 537, 539
polygons
calculating geometric data for 254
drawing 266, 276
polyface meshes 539
polyline arcs 276
polylines
calculating geometric data for 254
chamfering 350
closed 276
drawing 262
editing 353
exploding 354
filleting 346
grips 316, 355
joining 357
linetypes 185
modifying 293, 355
offsetting 334
polyline arcs 276
PostScript rendering 925
revision clouds 291
simplifying display 192
subobjects 356
wide polylines 263, 276
polysolids
drawing 475
portrait orientation 849, 864
PostScript files
exporting 925
PostScript fonts 744
precision
calculators 257
coordinate systems 199
cursor movements 232
distances 247
drawing units 57
Dynamic Input 223
geometric data 252
numeric values 795
object snaps 226
offset locations 241
point locations 241
predefined 3D meshes 540
preset 3D views 89
pressing areas of 3D solids 587
previewing
drawings 64, 72
layouts 64
object selections 301
plotted drawings 876
weblights 984
previous releases
converting files 928
fields and 736
lineweights and 190
saving drawings as 931
primitive solids
about 465
grips 567
modifying 590
spheres 471
primitives
mesh primitives 515
solids 465
printable areas 845, 848, 851, 861, 863
printers
offsetting plots 864
selecting 848, 861
printing
page setups 845
procedural surfaces
about 486
converting objects to 501
creating 442, 491
editing 594
profile tolerances 837
profiles (objects)
extruding 567
geometric objects used as 448
lofting 456
project lines 777
project sites
accessing Buzzsaw sites 947
creating 947
projected geometry 597
projected tolerances 840
prompts
dynamic prompts 225
properties
3D solids 590
blocks 416
colors 176
copying 165
dimension lines 775
fills 693
floating properties 416
hatches 693, 697
layers 165, 171
lighting 982
linetypes 180
lineweights 186
mass properties 256
meshes 514, 592
multiline properties 267
objects 161
overriding 173
plot styles 872
section objects 646
selecting objects by filtering 299
simplifying display 192
surfaces 489, 591
tables 757
text 712, 751
views 84
Properties palette
object properties 163
table properties 757
proxy fonts 741
proxy objects 937
proxy servers 943
PS files (PostScript)
exporting 925
publishing
Buzzsaw sites and 947
section objects 649
pulling areas of 3D solids 587
purging
block definitions 411
named objects 309
unreferenced linetypes 181
unused layers 170
pyramids
3D solids 472
mesh pyramids 521
modifying 590
Q
quadrant grips 315
quadrants in dimensioning 815
Quick Text mode 193
Quick View tools 64

R
radial dimensions 809, 822
radius (fillet) 345
radius dimensions 809, 833
RAL color sets 177
RAM
  allocating 105
  memory tuning 106
rapid decay area of spotlights 982
Rapid prototyping 927
raster images
  about 908
  attaching 911
  bitonal 916
  clipping boundaries 913
  display options 915
  display speed 920-921
  file formats 909
  file paths 911
  finding files 919
  hiding 921
  image information 917
  Internet-based files 911
loading 921
naming 919
pixels 908
preview images 72
resolution 912
scaling 912
tiled images 922
transparency 909
unloading 921
rays
  drawing 287
  filleting 347
  modifying 293
Readme help topic 5
Real face style 95
realistic rendering 993
Realistic visual style 93
realtime panning 81
rearranging
  objects in groups 306
reassociating dimensions 836
rebuilding surfaces 601
reconstructing surfaces 601
recording
  history of composite solids 582
recovering
  damaged drawings 75
recovery audits 76
rectangles
  drawing 265
  modifying 293
rectangular arrays 328
rectangular meshes 538
redoing actions 308
redrawing screen display 309
redundant edges 586
redundant faces 586
redundant vertices 586
reference parameters 391
reference points
  drawing 286
  offsetting from 248
referenced layers 170
references (xrefs) 881
refinement
  mesh objects 619, 628
  tessellation 513
refreshing screen display 309
regardless of feature size values 839
regenerating
  drawings 188
  layer display 167
  raster image display and 921
  screen display 193
regions
  calculating geometric data for 254
  composite 289
  creating 289
cross sections 639
relative coordinates
  2D coordinates 200
3D coordinates 206
copying objects with 326
cylindrical coordinates 208
entering 199
spherical coordinates 209
relative paths
xrefs 887–888
relaxing constraints 369
relocated xrefs 889
removing
3D solid history 582
back faces 994
blips 309
block attributes 435
block definitions 411
constraints 369
dimension breaks 824
edges 577
faces 574
fillets 346
geometric constraints 373
object definition 307
hidden surfaces 994
layer property overrides 174
multiline vertices 365
objects 310
objects from groups 306
points in splines 360
stray pixels 309
text styles 739
unreferenced linetypes 181
unused named objects 309
vertices 581
renaming
layers 171
Render (renderer) 1001
rendering
about 993
atmospheric effects 1001
background effects 1001
basic techniques 1002
configuring Renderer 1001
displaying images 1004
plotting rendered objects 868
preparing models for 994
saving images 1004
section objects 649
shaded viewport objects 854, 867
simplifying geometry for 996
repairing
damaged drawing files 75
replacing
dimension text 830
text 753
reshaping
objects 341, 587
resizing
3D solid faces 573
annotations 657, 659, 671
arrowheads 781
columns in text 722
dimension breaks 824
dimensions 801
edges 576
extending objects 337
fitting drawings on paper 852
layout viewports 147
leader objects 726
object grips and 314
objects 341
tables 758
text 750–751
trimming objects 337
resizing palettes 48
resolution
plotting 869
raster images 912
rendering and 999
restoring
backup files 77
damaged drawings 77
previous view 81
UCSs 211, 217
viewports 125
views 84
resulting color 963, 965, 983
resulting intensity 963, 965, 983
reversing actions 308
reviewing
revision clouds 291
revision clouds 291
revolved meshes 531
revolving solids 568
surfaces 568
revolving objects 459
RFS symbol (regardless of feature size) 839
right-hand rule 214
RLC files 908
rotated dimensions 807
rotating
  3D objects 324
  3D Rotate gizmo 549, 556
  3D subobjects 570
  3D views 90
  constraining rotation 557
dimension text 830
dimensions 807
edges 576
faces 573
grip methods for 314
hatches 698
lights 979
objects 313, 323
snap angles 235
UCSs 210
vertices 580
views in layout viewports 155
rotation snaps 318
rounding
corners 344
edges 577
rounding off
dimension values 795
drawing units 57
rows (tables)
  formatting 761
  modifying 757
RTF files 723
rubber-band lines
  locking 237
  tracking 246
ruled meshes 530
running object snaps 227, 230

S
S (regardless of feature size symbol) 839
SAT format files 907, 926
saving
  automatic saves 67, 77
  backup files 67, 77
cross sections 647
drawings 67, 944
files in older formats 928, 931
incremental saves 69
layout templates 157
objects 68
page setups 846
partial saves 68
preview images 72
rendered images 1004
selection filters 299
UCSs 217
viewports 125
views 84
visual fidelity and 68, 661, 928
scale
  annotations 657–658, 671
dimensions 793, 801
drawings 131
drawings 131
layout viewports 148
linetypes 184
multilines 267
plot scale 852, 865
scale factors 342
scale factors
  layout viewports 148
model space settings 132
raster images 912
scaling objects 342
scale locking in layout viewports 145
scaling
  3D Scale gizmo 549, 558
  3D subobjects 570
  annotations 657, 659, 671
  arrowheads 781
  blocks 407
  constraining scale 558
drawings 132
drawings to fit paper 852
edges 576
faces 573
grip methods for 314
hatch patterns 691, 698
linetypes 153, 184
lineweights 853
model space settings 131–132
objects 313, 342
plot scale 865
raster images 912
scaling by reference 343
tables 757–758
text 750–751
text scale ratios 133
vertices 580
views in layout viewports 148
scenes
lighting 960
rendering 1002
schedules
tables 757
screen display
cleaning up 309
clearing 44
performance 192
quality 921
updating 193
screening
layout viewports 151
scripts
customizing program startup 50
SDF files (space-delimited format) 429
search paths
default file locations 63
referenced drawings 888
specifying 73
searching
finding commands 11, 13, 22
finding drawing files 60, 72
finding raster images 919
text search and replace 753
section lines 636
section objects
about 635
cameras and 646
creating 637
grips 644
jogged segments 641
live sectioning 642
object states 645
plotting 650
properties 646
publishing 649
rendering 649
saving 647
views and 646
section planes 635
sectioning
2D sections 647
3D objects 647
3D views and 110
cameras and 646
jogged segments 641
live sectioning 635, 637, 642
lofting objects and 456, 568
publishing or plotting cross sections 649
section objects 635
views and 646
selecting
3D solid subobjects 562
classification properties and 299
CP and WP selection 296
customizing object selection default options for 302
deselecting 294
groups 305
highlighted objects 301
irregularly shaped selection areas 296
layers 170
methods for 297
multiple objects 295
objects 293
output devices 848, 861
overlapping objects 564
preventing object selection 298
previewing selections 301
selection fences 296
selection windows 295
subobject selection filters 565
suppressing highlighting 922
tables or table elements 759
selection fences 296
selection filters 565, 630
selection preview 294
selection windows 295
self-intersections 632
sending objects to back 196
servers
proxy servers 943
setting up pages 845
shaded images
AutoCAD LT functionality 935
customizing shading 95
derendering 999
visual styles 93
shaded UCS icon 221
shaded viewports
plotting options 854, 867–868, 870
resolution settings 869
Shaded visual style 93
Shaded with Edges visual style 93
shades (colors) 685
Shades of Gray visual style 93
shadows
hardware acceleration and 100
lighting and 96, 982
visual styles 100
shelling solids 585
SHX fonts 742, 744, 748
SI lighting units 959
silhouette edges 101
 single-line text
creating 708
editing 750
height 746
obliquing angle 747
orientation 748
styles 738
sketching freehand lines 269
Sketchy visual style 93
sky
natural lighting and 959
slanted text 747
slicing solids 483
 smooth lighting 96
smooth rendering 998
smoothing
mesh objects 615, 619, 628
tessellation 513
snapping (object snaps) 226
software acceleration 105
solid fills
creating 676, 685
overlapping objects 196
PostScript rendering 925
turning off 192
solid primitives
about 462
boxes 465
cones 468
creating 465
cylinders 469
pyramids 472
tori 474
wedges 466
solids
3D solids 477
composite solids 481
converting objects to 446
converting to meshes 534
creating 461
extruding 450
grips 567
interferences 484
lofting 456
modifying 547
polysolids 475
revolving 459
separating into original shapes 585
solid modeling 441
solid primitives 465
weeping 453
types of 447
source vectors for distant lights 984
space-delimited format files (SDF) 429
spacing
dimension elements 775
dimensions 825
equal intervals 248
grid lines 233
previous release formats and special characters
block attribute extraction files 429
bullets in lists 715
file names 69
fractions 721
inserting 711
tolerance stacks 721
Unicode text 945
xref symbols 900
spelling check 754
spheres
3D solids 471
mesh spheres 523
modifying 590
spherical coordinates 209
spinning mesh edges 613
spirals
drawing 285
modifying 363
spline-fit polylines
extending 340
trimming 340
splines
B-splines 502
calculating geometric data for 254
control points 360
drawing 281
editing 360
fit points 360
grips 316, 360
modifying 293
NURBS splines 504
offsetting 334
splitting
faces 628
mesh faces 622
spotlights
artificial lighting 960
attenuation rates 984
beam angle 982
falloff angle 982
field angle 982
hotspot angle 982
location 980
overview 965
rapid decay area 982
spreadsheets
attribute extraction data files 430
squares 266, 293
stacked text (multiline text) 720
Standard lighting workflow
about 959
distant lights 977
starting
customizing program startup 49
drawings 53
starting tables 760
STB files 859, 873
Stereolithograph (STL) files 927
Stereolithography Apparatus (SLA) 927
STL files 927
stray pixels 309
stretch-moves 321
stretching
grip methods for 314
objects 342
stretch-moves 321
styles
annotations 662
dimensions 773
leader objects 728
multiline styles 267
reference points 286
text 738
visual 93
subobjects 356, 570
substitute fonts 741, 744
subtracing
composite solids 481
subtracting
areas 255
objects 481
regions 290
sunlight
natural lighting 959–960
support files
search paths 73
surface models 442
surfaces
3D surfaces 485
about 486
associative 449, 505
blending 495
bulge magnitude 488
chamfering 577
composite solids 581
composite surfaces 584
constraints 508
continuity 488
converting meshes to 632
converting objects to 446, 501
converting to 3D solids 478, 480
converting to meshes 534
creating 486
drawings 65
gizmos 552
Swivel tool 109
symbol libraries
    creating 419
    inserting blocks 408
    updating block definitions 432
symbols
    arcs 818
    block attribute extraction data 429
    bullets in lists 715
    file names 69
    inserting 711
    material conditions 839
    projected tolerances 840
    symbols of termination 769
    tolerances 797
    xref symbols 900
symmetrical tolerances 797
syntax
    command line switches 50
T

table styles 760
Table toolbar 759
tables (inserted)
    annotations 655
    AutoFill feature 764
    breaking into parts 758
    cell styles 761
    column width 758
    creating 757
    defined 757
    editing 758
    fields in 733
    formatting 761
    formulas in 764
    gridlines 761
    grips 758
    inserting blocks in 762
    merging cells 759
previous release formats and 930

Index | 1073
<table>
<thead>
<tr>
<th>tiling</th>
<th>922</th>
</tr>
</thead>
<tbody>
<tr>
<td>images</td>
<td>922</td>
</tr>
<tr>
<td>tints</td>
<td>685</td>
</tr>
<tr>
<td>titles</td>
<td>761</td>
</tr>
<tr>
<td>tolerances</td>
<td></td>
</tr>
<tr>
<td>alignment</td>
<td>798</td>
</tr>
<tr>
<td>annotations</td>
<td>655, 665</td>
</tr>
<tr>
<td>composite tolerances</td>
<td>841</td>
</tr>
<tr>
<td>datum reference frames</td>
<td>840</td>
</tr>
<tr>
<td>deviation tolerances</td>
<td>797</td>
</tr>
<tr>
<td>geometric tolerances</td>
<td>837</td>
</tr>
<tr>
<td>inspection dimensions</td>
<td>821</td>
</tr>
<tr>
<td>lateral tolerances</td>
<td>797</td>
</tr>
<tr>
<td>limits</td>
<td>798</td>
</tr>
<tr>
<td>material conditions</td>
<td>839</td>
</tr>
<tr>
<td>multiline text</td>
<td>720</td>
</tr>
<tr>
<td>projected tolerances</td>
<td>840</td>
</tr>
<tr>
<td>rounding values</td>
<td>795</td>
</tr>
<tr>
<td>suppressing zeros</td>
<td>798</td>
</tr>
<tr>
<td>symbols</td>
<td>797</td>
</tr>
<tr>
<td>symmetrical tolerances</td>
<td>797</td>
</tr>
<tr>
<td>tone mapping</td>
<td>959</td>
</tr>
<tr>
<td>tools</td>
<td></td>
</tr>
<tr>
<td>precision drawing tools</td>
<td>199</td>
</tr>
<tr>
<td>tooltips</td>
<td></td>
</tr>
<tr>
<td>AutoSnap</td>
<td>229</td>
</tr>
<tr>
<td>Dynamic Input</td>
<td>223</td>
</tr>
<tr>
<td>dynamic prompts</td>
<td>225</td>
</tr>
<tr>
<td>settings</td>
<td>44</td>
</tr>
<tr>
<td>tori</td>
<td></td>
</tr>
<tr>
<td>3D solids</td>
<td>474</td>
</tr>
<tr>
<td>mesh tori</td>
<td>527</td>
</tr>
<tr>
<td>modifying</td>
<td>590</td>
</tr>
<tr>
<td>torus objects</td>
<td></td>
</tr>
<tr>
<td>3D solids</td>
<td>474</td>
</tr>
<tr>
<td>mesh tori</td>
<td>527</td>
</tr>
<tr>
<td>modifying</td>
<td>590</td>
</tr>
<tr>
<td>tracking</td>
<td></td>
</tr>
<tr>
<td>object snap tracking</td>
<td>244</td>
</tr>
<tr>
<td>polar tracking</td>
<td>239</td>
</tr>
<tr>
<td>specifying points</td>
<td>246</td>
</tr>
<tr>
<td>tracking points</td>
<td>244</td>
</tr>
<tr>
<td>training</td>
<td>4</td>
</tr>
<tr>
<td>transparency</td>
<td></td>
</tr>
<tr>
<td>hatches</td>
<td>693</td>
</tr>
<tr>
<td>layers</td>
<td>167, 194</td>
</tr>
<tr>
<td>objects</td>
<td>194</td>
</tr>
<tr>
<td>plot options</td>
<td>195, 870</td>
</tr>
<tr>
<td>raster images</td>
<td>909, 916</td>
</tr>
<tr>
<td>section planes</td>
<td>646</td>
</tr>
<tr>
<td>visual styles</td>
<td>97</td>
</tr>
<tr>
<td>triangles</td>
<td>266</td>
</tr>
<tr>
<td>trimming</td>
<td></td>
</tr>
<tr>
<td>3D objects</td>
<td>340</td>
</tr>
<tr>
<td>chamfered objects</td>
<td>349</td>
</tr>
<tr>
<td>filleted objects</td>
<td>345</td>
</tr>
<tr>
<td>hatches</td>
<td>694</td>
</tr>
<tr>
<td>objects</td>
<td>337</td>
</tr>
<tr>
<td>replacing trimmed areas</td>
<td>601</td>
</tr>
<tr>
<td>surfaces</td>
<td>596</td>
</tr>
<tr>
<td>True Color colors</td>
<td>176</td>
</tr>
<tr>
<td>TrueType fonts</td>
<td></td>
</tr>
<tr>
<td>about</td>
<td>741</td>
</tr>
<tr>
<td>PostScript font equivalents</td>
<td>744</td>
</tr>
<tr>
<td>proxy fonts</td>
<td>745</td>
</tr>
<tr>
<td>text height</td>
<td>746</td>
</tr>
<tr>
<td>vertical text</td>
<td>748</td>
</tr>
<tr>
<td>TXT files</td>
<td></td>
</tr>
<tr>
<td>importing</td>
<td>723</td>
</tr>
<tr>
<td>Type C goniometers</td>
<td>972</td>
</tr>
<tr>
<td>typefaces</td>
<td>740</td>
</tr>
</tbody>
</table>

**U**

| UCS icon | |
| coloring | 43 |
| display options | 221 |
| UCSs (user coordinate systems) | |
| ViewCube and | 119 |
| UCSs (user coordinate systems) | |
| 3D settings | 216 |
| 3D space and | 214 |
| about | 210 |
| assigning to viewports | 219 |
| AutoCAD LT functionality | 934 |
| defining | 210 |
| dynamic UCSs | 218 |
| elevations | 216 |
| multiple viewports | 216 |
paper space and 217
preset systems 216
restoring 217
saving 217
UCS icon 221
workplanes 214
unbounded hatches 675, 694
unconstrained drawings 368
underconstrained drawings 368
undoing actions
correcting mistakes 307
layer changes 171
Unicode fonts 742
Unicode text 711, 930, 945
unions
3D solids 481
regions 289
separating into original shapes 585
units of measurement
about 54
coordinate unit types 200
dimensions 790, 792
model space 131
plot scale 866
unloaded raster images 918
unloading
raster images 921
xrefs 905
unlocking
layers 167
unreferenced linetypes 181
untrimming surfaces 596
unused layers 170
updating
annotations 659, 671
associative dimensions 771
block attributes 435
block definitions 432
block references 435
display 193
fields 735
xrefs 881, 890
URLs (Uniform Resource Locators)
opening drawings with 944
saving drawings to 944
user coordinate system (UCS) 210
V
validating
3D solids 586
surfaces 602
values
dimension values 792
suppressing zeros 796
text fields 733
variable block attributes 421
vector expressions 258
verifying constraints 379
versions of AutoCAD 931
vertex editing 580
vertical dimension text 788
vertical dimensions 804
vertical text 748
vertices
3D solid vertices 570
creases 621
deleting 581
modifying 580
multilines 365
redundant vertices 586
rendering 998
surface control vertices 600–601
video cards 105
ViewCube 111
viewpoints in 3D space 90, 110
viewports
aligning views in 154
annotations in 657–658
arranging 122, 125
backgrounds in 99
creating 121
current 124
drawing in 124
layer property overrides and 173
layers and 173
layout viewports 145
lighting modes 955
plot options 854, 867
plotting 870
rendering 1002
shadows in 100
UCSs and 216, 219
visual styles in 94, 174
views
2D isometric 951
3D projection styles 85
3D views 107
aligning in viewports 154
changing 81
cross sections 635
flattened 3D objects 650
layout viewports 147
live sectioning 646
model space 3D views 89
modifying 144
multiple-view drawing layouts 137
panning 81
performance issues and 105
plot options 850
preset 3D views 89
properties 84
rendering 1002
restoring 84
rotating 155
saving 84
scaling 148
viewports 121
zooming 81
visibility
annotations 670
layers 144, 167
layout viewports and 149
transparency 194
xref layers 884
visual fidelity
annotative objects and 661
previous release file formats and 928
saving files and 68
visual styles
AutoCAD LT functionality 935
backgrounds 99
customizing 95
default styles 93
defined 93
edge display 101
face styles 95
layer property overrides and 174
lighting 94
modifying 94
performance issues and 105
plotting 868
shading 95
shadows 100
Visual Styles Manager 93
VIZ 1005
volumes 256

W
walls
drawing 475
WCS (world coordinate system)
about 210
weather 960
web folders
opening drawings from 944
weblights
about 967
distribution in photometric webs 969
free weblights 968
goniometric diagrams 968
luminous intensity distribution (LID) 972
properties 984
websites
Buzzsaw sites 947
wedges
3D solids 466
mesh wedges 525
modifying 590
wide polylines
drawing 263, 276
extending 339
simplified display 192
tapering segments 277
trimming 339
width
table columns 757
windows
interface options 43
selection windows 295
zooming to 82
Windows Clipboard 310
wipeout objects 702
Wireframe visual style 93
wireframes
    about 542
    plot options 854
    plotting 868
witness lines 777
workflows
    surface modeling 600
    workplanes 199, 214
world coordinate system (WCS) 210
WPolygon selection 296
wrinkled surfaces 601

X
X axis 43
X-ray visual style 93
X, Y coordinates 199
xlines 287
    filleting 347
    modifying 293
xrefs (external references) 881
XY planes (workplanes) 91, 199

Y
Y axis 43

Z
Z axis 43
zebra analysis 606
zero suppression
    dimensions 796
    tolerances 798
Zoom tool 109
zooming
    3D views 109
    about 81
    controlling zooming 82
    grid display and 234
    layout viewports 144
    lineweight display and 188