

AutoCAD®
2007

User's Guide

Autodesk®

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Contents

Chapter 1	Find the Information You Need	1
Install the Product	2	
Use the Help System Efficiently	3	
Find Information in Help	3	
Use Searches	5	
Understand How Help Topics Are Organized	8	
Use the Up Arrow to Navigate Help Topics	9	
Print Help Topics	9	
Show and Hide the Contents Pane	9	
Get Additional Help	10	
Use Quick Help on the Info Palette	10	
Learn the Product	12	
Access Subscription Center	13	
Overview of Subscription Center	13	
Integration of the Vault	15	
Receive Product Updates and Announcements	16	
Overview of Communication Center	16	
Customize the Update and Announcement Options	18	
Receive New Information Notifications	19	
View the Product Readme	20	
Part 1	The User Interface	21

Chapter 2	Toolbars and Menus	23
	Toolbars	24
	The Menu Bar	25
	Shortcut Menus	26
	The Dashboard	30
Chapter 3	The Command Window	33
	Enter Commands on the Command Line	34
	Enter System Variables on the Command Line	36
	Navigate and Edit Within the Command Window	36
	Switch Between Dialog Boxes and the Command Line	38
	Dock, Resize, and Hide the Command Window	39
Chapter 4	DesignCenter	43
	Overview of DesignCenter	44
	Understand the DesignCenter Window	44
	Access Content with DesignCenter	46
	Add Content with DesignCenter	49
	Retrieve Content from the Web with DesignCenter Online	54
	Overview of DesignCenter Online	54
	Understand DesignCenter Online Content Types	56
	Retrieve Content from the Web	57
Chapter 5	Customize the Drawing Environment	61
	Set Interface Options	62
	Create Task-Based Workspaces	69
	Save and Restore Interface Settings (Profiles)	72
	Customize Startup	74
Chapter 6	Tool Palettes	79
	Create and Use Tools from Objects and Images	80
	Create and Use Command Tools	86
	Change Tool Palette Settings	88
	Control Tool Properties	90
	Customize Tool Palettes	94
	Organize Tool Palettes	98
	Save and Share Tool Palettes	102
Part 2	Start, Organize, and Save a Drawing	105
Chapter 7	Start a Drawing	107
	Start a Drawing from Scratch	108
	Use a Wizard to Start a Drawing	110

Use a Template File to Start a Drawing	111
Specify Units and Unit Formats	114
Determine the Units of Measurement	114
Set Linear Unit Conventions	115
Set Angular Unit Conventions	118
Add Identifying Information to Drawings	119
Chapter 8 Open or Save a Drawing	121
Open a Drawing	122
Open Part of a Large Drawing (Partial Load)	123
Work with Multiple Open Drawings	126
Save a Drawing	127
Find a Drawing File	130
Specify Search Paths and File Locations	131
Chapter 9 Repair, Restore, or Recover Drawing Files	133
Repair a Damaged Drawing File	134
Create and Restore Backup Files	136
Recover from a System Failure	137
Chapter 10 Maintain Standards in Drawings	141
Overview of CAD Standards	142
Define Standards	144
Check Drawings for Standards Violations	146
Translate Layer Names and Properties	153
Convert Layers to Established Drawing Standards	153
View Selected Drawing Layers	155
Purge Unreferenced Layers	155
Part 3 Control the Drawing Views	157
Chapter 11 Change Views	159
Pan or Zoom a View	160
Pan and Zoom with the Aerial View Window	164
Save and Restore Views	167
Control the 3D Projection Style	171
Overview of Parallel and Perspective Views	171
Define a Perspective Projection (DVIEW)	172
Define a Parallel Projection	173
Choose Preset 3D Views	174
Define a 3D View with Coordinate Values or Angles	175
Change to a View of the XY Plane	177
Shade a Model and Use Edge Effects	178
Use a Visual Style to Display Your Model	178

Chapter 12	Use 3D Viewing Tools	183
Overview of 3D Views	?	
Use 3D Navigation Tools	?	
Walk and Fly Through a Drawing	?	
Create a 3D Dynamic View (DVIEW)	?	
Define a 3D View with a Camera	184	
Overview of Cameras	184	
Create a Camera	184	
Change Camera Properties	185	
Create Motion Path Animations	189	
Control a Camera Motion Path	189	
Specify Motion Path Settings	190	
Record a Motion Path Animation	191	
Chapter 13	Display Multiple Views in Model Space	193
Set Model Space Viewports	194	
Select and Use the Current Viewport	196	
Save and Restore Model Tab Viewport Arrangements	197	
Part 4	Choose a Work Process Before You Begin	201
Chapter 14	Create Single-View Drawings (Model Space)	203
Quick Start for Model Space Drafting	204	
Draw, Scale, and Annotate in Model Space	204	
Chapter 15	Create Multiple-View Drawing Layouts (Paper Space)	209
Quick Start for Layouts	210	
Understand the Layout Process	210	
Work with Model Space and Paper Space	211	
Work on the Model Tab	211	
Work on a Layout Tab	212	
Access Model Space from a Layout Viewport	217	
Create and Modify Layout Viewports	219	
Control Views in Layout Viewports	223	
Scale Views in Layout Viewports	223	
Control Visibility in Layout Viewports	225	
Scale Linetypes in Layout Viewports	231	
Align Views in Layout Viewports	233	
Rotate Views in Layout Viewports	235	
Reuse Layouts and Layout Settings	236	
Chapter 16	Work with Sheets in a Sheet Set	239
Quick Start for Sheet Sets	240	
Understand the Sheet Set Manager Interface	240	

Create and Manage a Sheet Set	242
Create a Sheet Set	242
Organize a Sheet Set	246
Create and Modify Sheets	249
Include Information with Sheets and Sheet Sets	259
Publish, Transmit, and Archive Sheet Sets	261
Use Sheet Sets in a Team	266
Part 5 Create and Modify Objects	269
Chapter 17 Control the Properties of Objects	271
Overview of Object Properties	273
Display and Change the Properties of Objects	273
Copy Properties Between Objects	276
Work with Layers	277
Overview of Layers	277
Use Layers to Manage Complexity	278
Create and Name Layers	287
Change Layer Settings and Layer Properties	290
Filter and Sort the List of Layers	293
Save and Restore Layer Settings	299
Work with Colors	302
Set the Current Color	302
Change the Color of an Object	305
Use Color Books	307
Work with Linetypes	309
Overview of Linetypes	309
Load Linetypes	310
Set the Current Linetype	312
Change the Linetype of an Object	313
Control Linetype Scale	315
Display Linetypes on Short Segments and Polylines	316
Control Lineweights	317
Overview of Lineweights	317
Display Lineweights	319
Set the Current Lineweight	321
Change the Lineweight of an Object	321
Control the Display Properties of Certain Objects	322
Control the Display of Polylines, Hatches, Gradient Fills, Lineweights, and Text	323
Control How Overlapping Objects Are Displayed	325
Chapter 18 Use Precision Tools	327
Use Coordinates and Coordinate Systems (UCS)	328
Overview of Coordinate Entry	328
Enter 2D Coordinates	330

Enter 3D Coordinates	334
Understand the User Coordinate System (UCS)	339
Specify Workplanes in 3D (UCS)	343
Assign User Coordinate System Orientations to Viewports	354
Control the Display of the User Coordinate System Icon	356
Use Dynamic Input	358
Snap to Locations on Objects (Object Snaps)	364
Use Object Snaps	364
The Object Snap Menu	367
Set Visual Aids for Object Snaps (AutoSnap)	367
Override Object Snap Settings	369
Restrict Cursor Movement	371
Adjust Grid and Grid Snap	371
Use Orthogonal Locking (Ortho Mode)	376
Use Polar Tracking and PolarSnap	377
Lock an Angle for One Point (Angle)	380
Combine or Offset Points and Coordinates	380
Combine Coordinate Values (Coordinate Filters)	381
Track to Points on Objects (Object Snap Tracking)	383
Track to Offset Point Locations (Tracking)	386
Specify Distances	387
Enter Direct Distances	387
Offset from Temporary Reference Points	387
Specify Intervals on Objects	388
Extract Geometric Information from Objects	391
Obtain Distances, Angles, and Point Locations	391
Obtain Area Information	392
Use a Calculator	397
Use the QuickCalc Calculator	397
Use the Command Line Calculator	414
Chapter 19 Draw Geometric Objects	417
Draw Linear Objects	418
Draw Lines	418
Draw Polylines	419
Draw Rectangles and Polygons	423
Draw Multiline Objects	426
Draw Freehand Sketches	429
Draw Curved Objects	431
Draw Arcs	432
Draw Circles	436
Draw Polyline Arcs	438
Draw Donuts	442
Draw Ellipses	443
Draw Splines	445
Draw Helixes	447
Draw Construction and Reference Geometry	449

Draw Reference Points	449
Draw Construction Lines (and Rays)	450
Create and Combine Areas (Regions)	452
Create Revision Clouds	455
Chapter 20 Create and Use Blocks (Symbols)	459
Overview of Blocks	460
Create and Store Blocks	460
How Blocks Are Stored and Referenced	460
Create Blocks Within a Drawing	461
Create Block Libraries	463
Create Drawing Files for Use as Blocks	464
Use Tool Palettes to Organize Blocks	467
Add Dynamic Behavior to Blocks	467
Quick Start to Creating Dynamic Blocks	467
Overview of Dynamic Blocks	468
Use the Block Editor	471
Add Dynamic Elements to Blocks	482
Save a Block in the Block Editor	551
Control the Color and Linetype Properties in Blocks	553
Nest Blocks	555
Insert Blocks	556
Work with Dynamic Blocks in Drawings	560
Attach Data to Blocks (Block Attributes)	563
Overview of Block Attributes	563
Define Block Attributes	564
Extract Data from Block Attributes	566
Extract Block Attribute Data (Advanced)	568
Modify Blocks	573
Modify a Block Definition	573
Modify Block Attributes	576
Change the Color and Linetype in a Block	581
Disassemble a Block Reference (Explode)	583
Remove Block Definitions	584
Chapter 21 Change Existing Objects	587
Select Objects	588
Select Objects Individually	588
Select Multiple Objects	590
Prevent Objects from Being Selected	592
Filter Selection Sets	593
Customize Object Selection	596
Group Objects	601
Correct Mistakes	605
Erase Objects	607
Use Windows Cut, Copy, and Paste	609
Modify Objects	611

Choose a Method to Modify Objects	611
Move or Rotate Objects	612
Copy, Offset, or Mirror Objects	620
Change the Size and Shape of Objects	631
Fillet, Chamfer, Break, or Join Objects	642
Use Grips to Edit Objects	654
Modify Complex Objects	664
Disassociate Compound Objects (Explode)	664
Modify or Join Polylines	665
Modify Splines	669
Modify Helixes	672
Modify Multilines	673
Part 6 Work with 3D Models	677
Chapter 22 Create 3D Models	679
Overview of 3D Modeling	680
Create 3D Solids and Surfaces	681
Overview of Creating 3D Solids and Surfaces	681
Create 3D Solid Primitives	682
Create a Polysolid	691
Create Solids and Surfaces from Lines and Curves	694
Create Solids and Surfaces from Objects	704
Create Solids from Surfaces	707
Create Composite Solids	707
Create Solids by Slicing	710
Check for Interferences within a Solid Model	711
Create Meshes	713
Create Wireframe Models	722
Add 3D Thickness to Objects	724
Chapter 23 Modify 3D Solids and Surfaces	727
Manipulate 3D Solids and Surfaces	728
Manipulate Individual Solids and Surfaces	728
Work with Composite Solids	731
Select 3D Subobjects	? ?
Move, Rotate, and Scale Subobjects	? ?
Modify Faces on 3D Solids	? ?
Modify Edges on 3D Solids	? ?
Modify Vertices on 3D Solids	? ?
Press or Pull Bounded Areas	734
Use Grip Tools to Modify Objects	735
Understand Grip Tools	735
Use the Move Grip Tool to Modify Objects	738
Use the Rotate Grip Tool to Modify Objects	741
Add Edges and Faces to Solids	743

Separate 3D Solids	744
Shell 3D Solids	745
Clean and Check 3D Solids	746
Chapter 24 Create Sections and 2D Drawings from 3D Models	749
Section 3D Solids	750
Work with Section Objects	751
Overview of Section Objects	751
Create Section Objects	751
Set Section Object States	754
Use Section Object Grips	754
Access Section Object Shortcut Menu	756
Change Section Object Properties	757
Associate Section Objects with Views and Cameras	758
Publish Section Objects	758
Save Section Object as Tool Palette Tool	759
Add Jogs to a Section	759
Live Sectioning	761
Understand Live Sectioning Behavior	762
Generate 2D and 3D Sections	763
Create a Flattened View	764
Part 7 Hatches, Notes, Tables, and Dimensions	767
Chapter 25 Hatches, Fills, and Wipeouts	769
Overview of Hatch Patterns and Fills	770
Define Hatch Boundaries	775
Overview of Hatch Boundaries	775
Control the Hatching in Islands	776
Define Hatch Boundaries in Large Drawings	778
Create Unbounded Hatches	779
Choose Hatch Patterns and Solid Fills	780
Create Solid-Filled Areas	780
Create Gradient-Filled Areas	783
Use Predefined Hatch Patterns	785
Create User-Defined Hatch Patterns	786
Modify Hatches and Solid-Filled Areas	787
Create a Blank Area to Cover Objects	790
Chapter 26 Notes and Labels	793
Overview of Notes and Labels	794
Create Text	794
Overview of Creating Text and Leaders	794
Create Single-Line Text	795
Create Multiline Text	798

Create Text with Leaders	817
Import Text from External Files	822
Use Fields in Text	823
Insert Fields	823
Update Fields	828
Use Hyperlinks in Fields	830
Work with Text Styles	831
Overview of Text Styles	831
Assign Text Fonts	833
Set Text Height	838
Set Text Obliquing Angle	839
Set Horizontal or Vertical Text Orientation	840
Change Text	841
Overview of Changing Text	841
Change Single-Line Text	842
Change Multiline Text	843
Change Text with a Leader	846
Change Text Scale and Justification	847
Check Spelling	848
Use an Alternate Text Editor	850
Overview of Using an Alternate Text Editor	850
Format Multiline Text in an Alternate Text Editor	851
Chapter 27 Tables	855
Create and Modify Tables	856
Work with Table Styles	860
Add Text and Blocks to Tables	864
Use Formulas in Table Cells	868
Chapter 28 Dimensions and Tolerances	875
Understand Basic Concepts of Dimensioning	876
Overview of Dimensioning	876
Parts of a Dimension	877
Associative Dimensions	878
Use Dimension Styles	880
Overview of Dimension Styles	880
Compare Dimension Styles and Variables	881
Control Dimension Geometry	883
Control Dimension Text	889
Control Dimension Values	899
Set the Scale for Dimensions	906
Create Dimensions	908
Create Linear Dimensions	908
Create Radial Dimensions	915
Create Angular Dimensions	920
Create Ordinate Dimensions	922
Create Arc Length Dimensions	923

Modify Existing Dimensions	925
Apply a New Dimension Style to Existing Dimensions	925
Override a Dimension Style	926
Modify Dimension Text	928
Modify Dimension Geometry	931
Change Dimension Associativity	934
Add Geometric Tolerances	936
Overview of Geometric Tolerances	936
Material Conditions	938
Datum Reference Frames	938
Projected Tolerance Zones	939
Composite Tolerances	939
Part 8 Plot and Publish Drawings	943
Chapter 29 Prepare Drawings for Plotting and Publishing	945
Quick Start to Preparing Drawings for Plotting and Publishing	946
Specify Page Setup Settings	946
Overview of Page Setup Settings	946
Select a Printer or Plotter for a Layout	948
Select a Paper Size for a Layout	950
Set the Plot Area of a Layout	954
Adjust the Plot Offset of a Layout	955
Set the Plot Scale for a Layout	956
Set the Lineweight Scale for a Layout	958
Select a Plot Style Table for a Layout	959
Set Shaded Viewport and Plot Options for a Layout	962
Determine the Drawing Orientation of a Layout	964
Use the Layout Wizard to Specify Layout Settings	965
Import PCP or PC2 Settings into a Layout	965
Create and Use Named Page Setups	966
Use Named Page Setups with Sheet Sets	971
Chapter 30 Plot Drawings	973
Quick Start to Plotting	974
Overview of Plotting	974
Use a Page Setup to Specify Plot Settings	978
Select a Printer or Plotter	980
Specify the Area to Plot	980
Set Paper Size	981
Position the Drawing on the Paper	984
Specify the Printable Area	984
Set the Position of the Plot	985
Set Drawing Orientation	985
Control How Objects Are Plotted	985
Set Plot Scale	985

Set Shaded Viewport Options	988
Set Options for Plotted Objects	990
Use Plot Styles to Control Plotted Objects	995
Use Color-Dependent Plot Style Tables	1001
Use Named Plot Style Tables	1002
Change Plot Style Settings	1008
Preview a Plot	1017
Plot Files to Other Formats	1019
Plot DWF Files	1019
Plot to DXB File Formats	1020
Plot to Raster File Formats	1021
Plot Adobe PDF Files	1022
Plot Adobe PostScript Files	1024
Create Plot Files	1025
Chapter 31 Publish Drawings	1027
Quick Start to Publishing	1028
Overview of Publishing	1028
Create and Modify a Drawing Set for Publishing	1031
Create a Paper or Plot File Drawing Set	1037
Publish an Electronic Drawing Set	1040
Publish a Sheet Set	1043
Republish a Drawing Set	1048
View a Published Electronic Drawing Set	1050
Set Publish Options	1050
3D DWF Publishing	1058
Configure a DWF6 Driver (Advanced)	1064
Overview of Creating or Modifying a DWF6 Configuration File	1065
Set the DWF File Resolution	1068
Set the DWF File Compression	1070
Set Font Handling for the DWF File	1071
Edit Pen Patterns for the DWF File	1072
Part 9 Share Data Between Drawings and Applications	1075
Chapter 32 Reference Other Drawing Files	1077
Overview of Referenced Drawings (Xrefs)	1078
Attach and Detach Referenced Drawings	1078
Attach Drawing References (Xrefs)	1078
Set Paths to Referenced Drawings	1082
Detach Referenced Drawings	1087
Update and Bind Referenced Drawings	1088
Nest and Overlay Referenced Drawings	1088
Update Referenced Drawing Attachments	1090

Archive Drawings That Contain Referenced Drawings	1091
(Bind)	1091
Understand Clipping Boundaries	1093
Clip Referenced Drawings and Blocks	1093
Define Clipping Boundaries	1094
Edit Referenced Drawings	1095
Edit a Referenced Drawing in a Separate Window	1096
Edit Selected Objects in Referenced Drawings and Blocks	1096
Use the Working Set to Edit Referenced Drawings and Blocks	1099
Save Back Edited Referenced Drawings and Blocks	1101
Edit Referenced Drawings and Blocks with Nesting, OLE, or Attributes	1102
Resolve Referenced Drawing Errors	1103
Resolve Missing External References	1103
Resolve Circular External References	1105
Resolve Name Conflicts in External References	1105
Track External Reference Operations (Log File)	1107
Increase Performance with Large Referenced Drawings	1109
Overview of Demand Loading	1109
Unload Xrefs	1109
Work with Demand Loading	1110
Work with Layer and Spatial Indexes	1111
Set Paths for Temporary Xref File Copies	1112
Part 10 Create Realistic Images and Graphics	1115
Chapter 33 Add Lighting to Your Model	1117
Overview of Lighting	1118
Create and Manipulate Lights	1118
Choose the Type of Light to Use	1118
Control the Location and Properties of Lights	1122
Simulate Sunlight	1127
Index	1131

Find the Information You Need

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

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

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

In this chapter

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

Install the Product

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

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

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

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

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

Migration Tools

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

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

Available tools include

- Layer State Converter
- DWG TrueConvert
- AutoLISP Compatibility Analyzer
- ScriptPro

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

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

Use the Help System Efficiently

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

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

Find Information in Help

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

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

Contents Tab

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

Index Tab

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

Search Tab (Microsoft Search)

- Provides full-text search of all the topics listed on the Contents tab.
- Allows you to perform an exhaustive search for a specific word or phrase.
- Displays a ranked list of topics that contain the word or words entered in the keyword field.

Search Tab (Natural-Language Search)

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

NOTE Depending on the language version of the product, the Search tab provides either the traditional Microsoft search that locates keywords and exact phrases, or a natural-language search that analyzes phrases or questions.

To start Help

- Click Help menu ► Help, or press F1.
For Developer Help, click Help ► Additional Resources ► Developer Help.

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

To use the Help Contents

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

Use Searches

Use the Search tab to find relevant topics based on a word or phrase that you enter.

Depending on the language version of the product, the Search tab provides either the traditional Microsoft search that locates keywords and exact phrases, or a natural-language search that analyzes phrases or questions.

You can tell which version that is used by your product by the background color and prompt.

- Natural-language search uses a blue background color and prompts you to enter a question.
- Traditional Microsoft search uses a gray background color and prompts you to enter one or more words.

Natural Language Search Tab

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

You can perform this natural-language query by entering a question, a word, or a phrase. For example, you could enter: **How do I create a layout?**

If the information that you are searching for is not listed in the first few topics, try rephrasing your question. You can also expand your natural language query to the Web.

Traditional Microsoft Search Tab

The basic search rules are as follows:

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

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

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

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

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

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

To search for information in Help (natural-language search)

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

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

To expand your queries to the Web (natural-language search)

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

A web search engine displays the results of your search.

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

To search for information in Help (Microsoft search)

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

- Click



to add Boolean operators to your search.

- Enter the word or phrase you want to appear after the Boolean operator.

(Optional) Further refine your search with the check boxes at the bottom of the Search tab:

- **Search Previous Results.** Refines a search by applying new search criteria to the results of a previous search.
- **Match Similar Words.** Expands the search to include words similar to your search criteria, rather than limiting the search to exact matches.
- **Search Titles Only.** Limits the search to topic titles, rather than searching titles and topic contents.

- 3 Click List Topics and select the topic you want. Click Display.
- 4 To sort the topic list, click the Title, Location, or Rank column heading.

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

Understand How Help Topics Are Organized

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

The tabs display different types of information.

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

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

Concepts Tab Organization

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

Procedures Tab and Commands Tab Organization

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

To change the type of Help information displayed

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

Use the Up Arrow to Navigate Help Topics

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

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

Print Help Topics

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

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

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

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

To print a Help topic

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

To print a selected heading and all subtopics

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

Show and Hide the Contents Pane

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

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



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



Get Additional Help

You can access several additional sources of help.

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

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

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

Use Quick Help on the Info Palette

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

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

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

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

To display Quick Help

- Click Help menu ► Info Palette.

Command line: ASSIST

To navigate Quick Help

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

Command line: ASSIST

To print Quick Help information

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

To lock Quick Help on the current topic

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

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

Command line: ASSIST

Learn the Product

Autodesk Authorized Training Centers

The Autodesk® Authorized Training Center (ATC®) network delivers Autodesk-authorized, instructor-led training to design professionals who use Autodesk software. Autodesk Authorized Training Centers use experienced and knowledgeable instructors. More than 1,200 ATC sites are available worldwide to meet your needs for discipline-specific, locally based training.

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

Autodesk Official Training Courseware

Autodesk Official Training Courseware (AOTC) is technical training material developed by Autodesk. Designed for traditional 1/2-day to 5-day, instructor-led classroom training and used by Authorized Training Centers and other Autodesk partners, AOTC is well-suited for self-paced, standalone learning. The manuals cover key concepts and software functionality with hands-on, step-by-step, real-world exercises. You can purchase AOTC from your local reseller or distributor, or you can order it online from the Autodesk Store at <http://www.autodesk.com/aotc>.

e-Learning

Autodesk e-Learning for Autodesk Subscription customers features interactive lessons organized into product catalogs. Each lesson is 20-40 minutes in length and features hands-on exercises, with an option to use a simulation or the software application. You can also use an online evaluation tool that identifies gaps in skills, determines what lessons will be most helpful, and gauges learning progress.

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

Autodesk Developer Network

The Autodesk Developer (ADN) program for ADN members provides support for full-time, professional developers who want to build software based on Autodesk products. As an ADN member, you will receive the business, software, support, and training you need to be successful. If you are a developer, visit <http://www.autodesk.com/adn>.

Autodesk Consulting

Autodesk Consulting provides services that help set up processes and provide critical training that will help increase productivity so you can capitalize on the power of your products. For more information on general consulting, systems integration, or custom training services, visit
<http://www.autodesk.com/consulting>.

Partner Products and Services

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

Access Subscription Center

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

Overview of Subscription Center

With Autodesk Subscription, you get the latest releases of Autodesk software, incremental product enhancements, personalized web support from Autodesk technical experts, and self-paced e-Learning. Subscription services are available to subscription members only.

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

- **Subscription e-Learning Catalog.** Features interactive lessons organized into product catalogs. Each lesson is 15-30 minutes and features hands-on exercises, with an option to use a simulation instead of the software application. You can use an online evaluation tool that identifies gaps in skills, determines what lessons will be most helpful, and gauges learning progress.
- **Create Support Request.** Provides direct one-to-one communication with Autodesk support technicians. You receive fast, complete answers to your installation, configuration, and troubleshooting questions.
- **View Support Requests.** Allows you to track and manage your questions and responses through Autodesk's state-of-the-art support system.

- **Edit Subscription Center Profile.** Allows you to set up and maintain your subscription account.

Subscription Resources and Privacy

Subscription resources provide interactive product features over the Internet. Each time you access subscription resource (such as e-Learning or Create Support Request) from the Help menu in an Autodesk product, product information (such as the serial number, version, language, and the subscription contract ID) is sent to Autodesk for verification that your product is on subscription.

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

Enable Subscription Resources

The CAD Manager Control utility can be used to turn Subscription Resources on and off in the Help menu. For more information about how to install the utility, see "To install the Subscription Center (part of the CAD Manager Control utility)." For more information about using the utility once it is installed, click Help in the CAD Manager Control utility window.

To access the Subscription Center in the program

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

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

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

- 1 Double-click *setup.exe* on the product CD.
- 2 In the Media Browser, Install tab, click either Multi-Seat Stand-Alone Installation or Network Deployment.
- 3 Under Install Supplemental Tools, click Autodesk CAD Manager Tools 4.0.
- 4 Under Autodesk CAD Manager Tools 4.0, click Install.

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

Integration of the Vault

If you are a Subscription customer, the Autodesk Vault, a file management tool, is a repository where documents and files are stored and managed. The Vault gives you more power to manage files and track changes. Versioned copies of master files are maintained, allowing you to easily revert to earlier versions of files. You can check files out for editing and later check them back in. The master copy is never directly edited. During the check-in process, you can add comments regarding the edits you've made to inform other designers of your changes. You are able to quickly understand a project's developmental flow and history.

The Autodesk Vault consists of two required components: the Autodesk Data Management Server and the Vault Client. Optionally, you can also install the Vault Office Add-in.

For information about using the Vault, refer to the Vault Help system.

NOTE The main components for the Autodesk Vault can be downloaded from the Autodesk Subscription site.

The Autodesk Data Management Server

The Autodesk Data Management Server stores the master copies of all your documents and designs. By storing all your data in a common, centralized location, you can easily share and manage information with your design team.

The Vault Client

The Vault Client software includes Autodesk Vault Explorer. The Vault Explorer is a stand-alone application that provides tools you use to access data stored on the Autodesk Data Management Server. From within AutoCAD, you can log in, log out, and open and attach files directly from the Vault. You can also access the Vault using the Vault Explorer. With the Vault Explorer you can

- 1 Manage the Vault
- 2 Manage Vault user accounts
- 3 Set working folders
- 4 Create folders in the Vault
- 5 Add, check in, and check out files (including non-Inventor and non-DWG files; exception is AutoCAD Electrical files)

- 6** Move files in the Vault
- 7** Rename files in the Vault
- 8** View the history of design changes
- 9** Create file associations within the Vault by attaching two or more files
- 10** Check out the latest version of a file
- 11** Package files using Pack and Go

The Vault Office Add-in (Optional)

This optional component gives Vault access to Microsoft® Office products (Word, Excel). The Vault access is similar to the access to Microsoft Office that is available in AutoCAD. To use this component, the Vault Client must be installed.

Receive Product Updates and Announcements

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

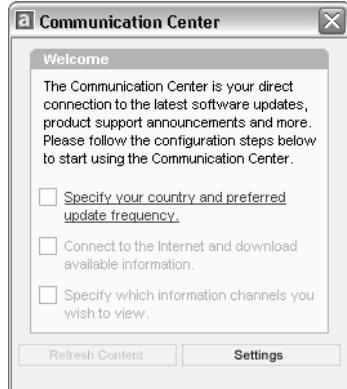
Overview of Communication Center

Communication Center provides the following kinds of announcements:

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

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

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



Communication Center Online Privacy

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

The following information is sent to Autodesk:

- **Product Name.** The name of the product in which you are using Communication Center
- **Product Release Number.** The version of the product
- **Product Language.** The language version of your product
- **Country/Region.** The country/region that was specified in the Communication Center settings
- **Your Subscription Contract Number.** The information sent to Autodesk if you entered it in the Error Report dialog box

Autodesk compiles statistics using the information sent from Communication Center to monitor how it is being used and how it can be improved. Autodesk will maintain information provided by or collected from you in accordance

with the company's published privacy policy, which is available on <http://www.autodesk.com/privacy>.

Turn Communication Center On or Off

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

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

- 1 Double-click *setup.exe* on the product CD.
- 2 In the Media Browser, Install tab, click either Multi-Seat Stand-Alone Installation or Network Deployment.
- 3 Under Install Supplemental Tools, click Autodesk CAD Manager Tools 4.0.
- 4 Under Autodesk CAD Manager Tools 4.0, click Install.

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

Customize the Update and Announcement Options

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

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

To customize Communication Center options

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

Receive New Information Notifications

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



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



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

To open the Communication Center window

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

View the Product Readme

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

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

- View the Readme

Part I

The User Interface

Chapter 2 Toolbars and Menus

Chapter 3 The Command Window

Chapter 4 DesignCenter

Chapter 5 Customize the Drawing Environment

Chapter 6 Pointing Devices

2

Toolbars and Menus

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

In this chapter

- Toolbars
- The Menu Bar
- Shortcut Menus
- The Dashboard

Toolbars

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

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

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

Display or Hide, Dock, and Resize Toolbars

AutoCAD initially displays several toolbars such as

- Standard toolbar
- Styles toolbar
- Layers toolbar

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

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

See also:

- [Create Task-Based Workspaces](#)
- [“Customize Toolbars” in the *Customization Guide*](#)

To display a toolbar

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

To dock a toolbar

- 1 Position the cursor on the name of the toolbar or in any blank area, and hold down the button on your pointing device.

- 2** Drag the toolbar to a docking location at the top, bottom, or either side of the drawing area.
- 3** When the outline of the toolbar is displayed in the docking area, release the button.
To place a toolbar in a docking region without docking it, hold down CTRL as you drag.

To undock a toolbar

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

To resize a toolbar

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

To close a toolbar

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

The Menu Bar

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

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

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

See also:

["Create Pull-Down and Shortcut Menus" in the *Customization Guide*](#)

To use a menu

Choose one of the following methods:

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

Shortcut Menus

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

You can display different shortcut menus when you right-click different areas of the screen, including

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

Shortcut menus typically include options to

- Repeat the last command entered
- Cancel the current command
- Display a list of recent user input
- Cut, copy, and paste from the Clipboard
- Select a different command option

- Display a dialog box, such as Options or Customize
- Undo the last command entered

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

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

See also:

["Create Submenus" in the *Customization Guide*](#)

To display a shortcut menu

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

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

To turn off shortcut menus in the drawing area

- 1 Click Tools menu ▶ Options.
- 2 In the Options dialog box, User Preferences tab, under Windows Standard Behavior, clear Shortcut Menus in Drawing Area.
- 3 To control Default, Edit, and Command shortcut menus individually, select Shortcut Menus in Drawing Area. Right-Click Customization.
- 4 In the Right-Click Customization dialog box, under Default Mode or Edit Mode, select one of the following options to control what happens when you right-click in the drawing area and no command is in progress:
 - **Repeat Last Command.** Repeats the last command. Selecting this option turns off the Default and Edit shortcut menus. Right-clicking is the same as pressing ENTER.
 - **Shortcut Menu.** Displays the Default or Edit shortcut menu.

- 5 Under Command Mode, select one of the following options to determine what happens when you right-click in the drawing area while a command is in progress:
 - **Enter**. Turns off the Command shortcut menu. Right-clicking is the same as pressing ENTER.
 - **Shortcut Menu: Always Enabled**. Displays the Command shortcut menu.
 - **Shortcut Menu: Enabled When Command Options Are Present**. Displays the Command shortcut menu only when options are currently available in the Command prompt. In a Command prompt, options are enclosed in square brackets. If no options are available, right-clicking is the same as pressing ENTER.

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

To turn on time-sensitive right-click behavior

- 1 Click Tools menu > Options.
- 2 In the Options dialog box, User Preferences tab, under Windows Standard Behavior, click Right-Click Customization.
- 3 In the Right-Click Customization dialog box, select Turn on Time-Sensitive Right-Click.
You can specify the duration of the longer click. The default is 250 milliseconds
- 4 Click Apply & Close.
- 5 In the Options dialog box, click OK.

Command line: OPTIONS

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

- 1 Click the arrow at the right end of the status bar, and then click Tray Settings.

- 2 In the Tray Settings dialog box, select or clear the following display options:

 - **Display Icons from Services.** Displays the tray at the right end of the status bar and displays icons from services. When this option is cleared, the tray is not displayed.
 - **Display Notifications from Services.** Displays notifications from services such as Communications Center. When the Display Icons from Services option is cleared, this option is unavailable.
- 3 If Display Notifications from Services is selected, set a time for a notification to be displayed, or select Display Until Closed.
- 4 Click OK.

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

Command line: TRAYSETTINGS

To control the display of buttons on the status bar

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

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

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

To control the display of coordinates on the status bar

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

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

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

To control the display of recent input

- 1 At the Command prompt, enter **inpushistorymode**.
- 2 Enter a sum of one or more of the following values:

 - **0.** No history of recent input is displayed.

- **1.** History of recent input is displayed at the command line with access through Up Arrow and Down Arrow keys.
- **2.** History of recent input for the current command is displayed in the shortcut menu.
- **4.** History of recent input for all commands in the current session is displayed in the shortcut menu.
- **8.** Markers for recent input of point locations are displayed in the drawing.

The default value is 15.

- 3** (Optional) At the Command prompt, enter **cmdinpushistorymax**.
- 4** Enter a value to control how many unique values entered at a prompt are remembered and available to be displayed as recent input.

The Dashboard

The dashboard is a special palette that displays buttons and controls that are associated with a task-based workspace. It is used primarily for 3D modeling, viewing, and rendering.

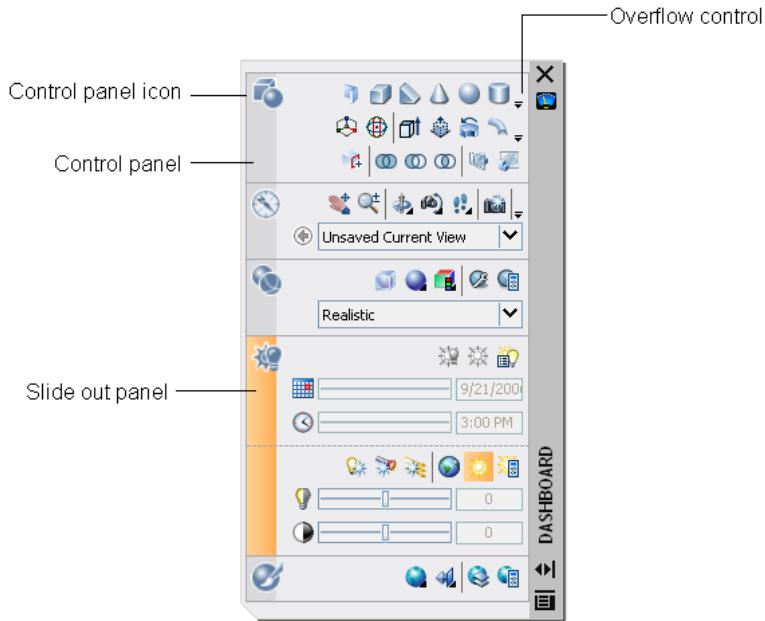
Overview

The dashboard provides a single interface element for modeling and rendering operations. It eliminates the need to display a lot of toolbars and reduces the clutter in the application window. Thus, you can maximize the area available for 3D work, and use a single interface for speed and convenience.

The dashboard displays automatically when you use the 3D Modeling workspace. You can display the dashboard manually by clicking the Tools ► Palettes menu or by entering DASHBOARD at the Command prompt.

Dashboard Organization and Operation

The dashboard is organized into a series of control panels. Each control panel includes related tools and controls that are similar to the tools in toolbars and controls in dialog boxes.



The large icons displayed at the left side of the dashboard are called control panel icons. Each control panel icon identifies the purpose of the control panel and, if you click the icon, opens a slide-out panel that contains additional tools and controls. An open slide-out panel closes automatically when you click another control panel icon. Only one slide-out panel is displayed at a time.

A tool palette group can be associated with each control panel. To display the associated tool palette group, click a tool or open a slide-out panel.

Customize the Dashboard

You can customize the dashboard in the following ways:

- You can control whether individual control panels are displayed by right-clicking the dashboard and, on the shortcut menu, clicking or clearing the names of control panels.
- You can associate a customizable tool palette group with each tool panel on the dashboard. Right-click the control panel to display a list of available tool palette groups.
- You can size the dashboard horizontally. If not enough space is available to display all the tools in a row, a black down-arrow called an overflow control, displays.

- You can control the auto-hide, docking, and anchoring behavior on the dashboard shortcut menu.

Anchoring the dashboard results in the dashboard being displayed as a narrow, space-saving strip along the left or right edge of the drawing area. For more information about docking and anchoring palettes and windows, see Set Interface Options.

To display the dashboard

- Click Tools menu > Palettes > Dashboard.

Command line: DASHBOARD

To change the dashboard rollover behavior

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

Rollover behavior is on.



Rollover behavior is off.



NOTE Rollover behavior is available only when the Dashboard window is undocked or when it is anchored.

To display or hide a control panel

- Right-click anywhere inside the Dashboard window. On the shortcut menu under Control Panels, click or clear the name of a control panel.

To display or hide a slide-out panel

- Click a control panel icon on the left side of the dashboard.

To associate a tool palette group with a control panel

- 1 Right-click a control panel icon on the dashboard.
- 2 Click Tool Palette Group and click an available tool palette group from the list.

3

The Command Window

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

In this chapter

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

Enter Commands on the Command Line

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

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

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

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

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

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

Specify Command Options

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

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

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

Execute Commands

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

Repeat and Cancel Commands

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

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

Command: **multiple circle**

To cancel a command in progress, press ESC.

Interrupt a Command with Another Command or System Variable

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

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

Command: **line**

Specify first point: **'grid**

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

Resuming LINE command

Specify first point:

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

See also:

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

To copy a command you have recently used

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

Enter System Variables on the Command Line

System variables are settings that control how certain commands work.

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

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

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

To change the setting of a system variable

- 1 At the Command prompt, enter the system variable name. For example, enter **gridmode** to change the grid setting.
- 2 To change the status of GRIDMODE, enter **1** for on or **0** for off. To retain the current value of the system variable, press ENTER.

To see a complete list of system variables

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

Command line: SETVAR

Navigate and Edit Within the Command Window

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

Use the standard keys:

- UP, DOWN, LEFT ARROW, and RIGHT ARROW

- INS, DEL
- PAGE UP, PAGE DOWN
- HOME, END
- BACKSPACE

You can repeat any command used in the current session by cycling through the commands in the command window with UP ARROW and DOWN ARROW and pressing ENTER. By default, pressing CTRL+C copies highlighted text to the Clipboard. Pressing CTRL+V pastes text from the Clipboard to the text window or the command window.

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

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

Use the Text Window

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

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

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

See also:

- [“Dock, Resize, and Hide the Command Window” on page 39](#)
- [Use Dynamic Input](#)

To display the text window

- From within the drawing area, press F2.

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

Command line: TEXTSCR

To close the text window

- From within the text window, press F2.

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

Command line: GRAPHSCR

To copy text from the text window to the command line

- 1 If the text window is not displayed, press F2 to display it.
- 2 Select the text you want to copy.
- 3 Right-click in the command window or text window. Click Paste to Command Line.

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

Command line: COPYCLIP, PASTECLIP

Switch Between Dialog Boxes and the Command Line

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

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

between the options in the dialog box and those available on the command line.

These system variables also affect the display of dialog boxes:

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

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

To use the command line version of a command

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

Dock, Resize, and Hide the Command Window

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

Dock the Command Window

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

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

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

Anchoring the Command Window

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

Resize the Command Window

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

Hide the Command Window

Hide the command line by clicking the Tools menu ► Command Line (or press CTRL+9). When you hide the command line, you can still enter commands. However, some commands and system variables return values at the command line, so you may want to display the command line in those instances. To display the command line when it's hidden, click the Tools menu ► Command Line (or press CTRL+9).

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

To float the command window

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

To make the floating command window transparent

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

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

To dock the command window

- Click the title bar and drag the command window until it is over the top or bottom docking region of the AutoCAD window. When the command window becomes the same width as the AutoCAD window, release the mouse button to dock it.

The docking region is an edge of the AutoCAD application window that allows you to dock a toolbar, palette, or the command window.

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

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

To resize the command window when it is docked

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

To hide the command window

- On the Tools menu, click Command Line.

NOTE Some commands and system variables return values at the command line, so you may want to display the command line in those instances. To display the command line when it's hidden, press CTRL+9 or click the Tools menu ► Command Line.

4

DesignCenter

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

In this chapter

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

Overview of DesignCenter

With DesignCenter, you can

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

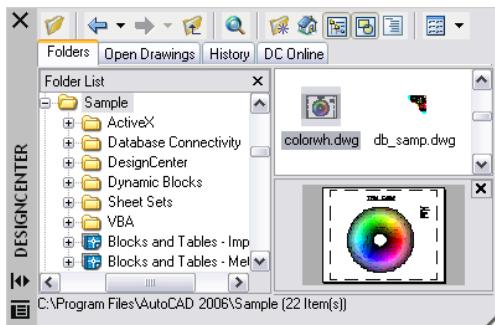
Understand the DesignCenter Window

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

The Organization of the DesignCenter Window

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

Undocked, the DesignCenter window is displayed as shown.



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

Control the Size, Location, and Appearance of DesignCenter

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

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

The DesignCenter Toolbar

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

To change the DesignCenter rollover behavior

- 1 If DesignCenter is not already open, click Tools menu ► DesignCenter.
- 2 Right-click the DesignCenter title bar. Click Auto-hide.

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

Standard toolbar



Command line: ADCENTER

To prevent DesignCenter from docking

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

Standard toolbar



Command line: ADCENTER

To display and hide the DesignCenter tree view

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

Standard toolbar



Command line: ADCENTER

Shortcut menu: Right-click the content area background and click Tree.

Access Content with DesignCenter

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



Folders Tab

The Folders tab displays a hierarchy of navigational icons, including

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

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

Open Drawings, History, and DC Online Tabs

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

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

Bookmark Frequently Used Content

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

When you select a drawing, folder, or another type of content and choose Add to Favorites, a shortcut to that item is added to the *Favorites* folder. The

original file or folder doesn't actually move; in fact, all the shortcuts you create are stored in the *Favorites* folder. The shortcuts saved in the *Favorites* folder can be moved, copied, or deleted using Windows® Explorer.

To change the source of the content displayed in DesignCenter

- 1 If DesignCenter is not already open, click Tools menu ► DesignCenter.
- 2 On the DesignCenter window, click one of the following tabs:
 - **Folders**. Lists your local and network drives.
 - **Open Drawings**. Lists the drawings that are currently open.
 - **History**. Lists the last 20 locations accessed through DesignCenter.
 - **DC Online**. Displays online content from the Web.

Standard toolbar



Command line: ADCENTER

To change the folder of the Home button in DesignCenter

- 1 If DesignCenter is not already open, click Tools menu ► DesignCenter.
- 2 In the DesignCenter tree view, navigate to the folder that you want to set as home.
- 3 Right-click on the folder. Click Set as Home.

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

Standard toolbar



Command line: ADCENTER

To add items to the *Favorites* folder in DesignCenter

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

Standard toolbar



Command line: ADCENTER

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

- 1 If DesignCenter is not already open, click Tools menu ► DesignCenter.
- 2 In DesignCenter, click the Favorites button.

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

Standard toolbar



Command line: ADCENTER

To organize your DesignCenter *Favorites* folder

- 1 If DesignCenter is not already open, click Tools menu ► DesignCenter.
- 2 In DesignCenter, click the Favorites button.
- 3 Right-click the background in the content area. Click Organize Favorites.
Your Autodesk *Favorites* folder is opened in a window.

Standard toolbar



Command line: ADCENTER

Add Content with DesignCenter

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

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

Add Content to a Drawing

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

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

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

Update Block Definitions with DesignCenter

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

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

Open Drawings with DesignCenter

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

Add Items from DesignCenter to a Tool Palette

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

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

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

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

To create a tool palette containing DesignCenter content

- 1 If DesignCenter is not already open, click Tools menu ► DesignCenter.
- 2 Do *one* of the following:
 - Right-click an item in the DesignCenter tree view. Click Create Tool Palette. The new tool palette contains the drawings, blocks, or hatches from the item you selected.
 - Right-click the background in the DesignCenter content area. Click Create Tool Palette. The new tool palette contains the drawings, blocks, or hatches from the DesignCenter content area.
 - Right-click a drawing in the DesignCenter tree view or content area. Click Create Tool Palette of Blocks. The new tool palette contains the blocks from the drawing you selected.

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

Standard toolbar



Command line: ADCENTER

To load the content area from the DesignCenter Search dialog box

- 1 If DesignCenter is not already open, click Tools menu ► DesignCenter.
- 2 In DesignCenter, use one of the following methods:
 - Drag the item from the search results list into the content area.
 - Double-click the item in the search results list.
 - Right-click the item in the search results list. Click Load into Content Area.
- 3 In the DesignCenter content area, double-click the Blocks icon.

Standard toolbar



Command line: ADCENTER

To load the content area of DesignCenter with a symbol library

- 1 If DesignCenter is not already open, click Tools menu ► DesignCenter.
- 2 On the DesignCenter toolbar, click Home.
- 3 In the content area, double-click the symbol library drawing that you want to load into the DesignCenter and then double-click the Blocks icon.

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

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

Standard toolbar



Command line: ADCENTER

To load the content area of DesignCenter with hatch patterns

- 1 If DesignCenter is not already open, click Tools menu ► DesignCenter.
- 2 On the DesignCenter toolbar, click Search.
- 3 In the Search dialog box, click the Look For box. Click Hatch Pattern Files.
- 4 On the Hatch Pattern Files tab, in the Search for the Name box, enter *.
- 5 Click Search Now.
- 6 Double-click one of the hatch pattern files that was found.

The hatch pattern file you selected is loaded into DesignCenter.

Standard toolbar



Command line: ADCENTER

To open a drawing from DesignCenter

- 1 If DesignCenter is not already open, click Tools menu ► DesignCenter.
- 2 In DesignCenter, do one of the following:
 - Right-click the drawing icon in the DesignCenter content area. Click Open in Application Window.
 - Press CTRL and drag the drawing icon from the DesignCenter content area to the drawing area of the application window.
(If you drag the drawing icon into the drawing area, a block is created in the current drawing.)
 - Drag the drawing icon from the DesignCenter content area to a location anywhere outside the drawing area of the application window.
(If you drag the drawing icon into the drawing area, a block is created in the current drawing.)

Standard toolbar



Command line: ADCENTER

To update a block definition with DesignCenter

- 1 If DesignCenter is not already open, click Tools menu ► DesignCenter.
- 2 Right-click a block in the DesignCenter content area. Click Redefine Only or Insert and Redefine.

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

Standard toolbar



Command line: ADCENTER

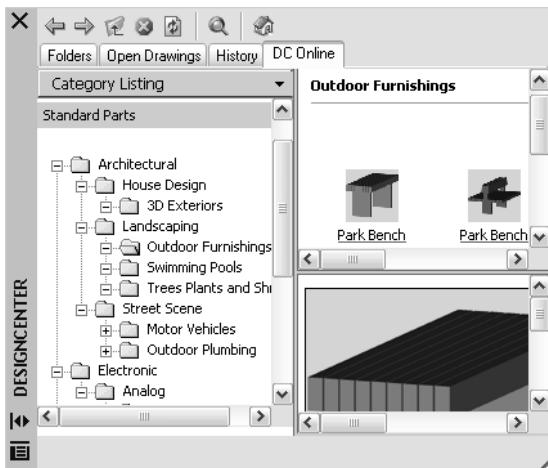
Retrieve Content from the Web with DesignCenter Online

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

Overview of DesignCenter Online

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

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

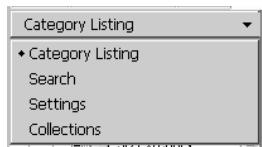


In the DesignCenter Online window, two panes are displayed—a right pane and a left pane. The right pane is called the *content area*. The content area displays the items or folders that you selected in the left pane. The left pane can display one of the following four views:

- **Category Listing.** Displays folders containing libraries of standard parts, manufacturer-specific content, and content aggregator websites.
- **Search.** Searches for online content. You can query items with Boolean and multiple-word search strings.

- **Settings.** Controls how many categories and items are displayed on each page in the content area as a result of a search or folder navigation.
- **Collections.** Specifies the discipline-specific content types that are displayed in DesignCenter Online.

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



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

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

DesignCenter Online Privacy

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

The following information is sent to Autodesk:

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

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

Turn the DC Online Tab On or Off

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

To install the CAD Manager Control utility

- 1 Double-click *setup.exe* on the product CD.
- 2 In the Media Browser, Install tab, click either Multi-Seat Stand-Alone Installation or Network Deployment.
- 3 Under Install Supplemental Tools, click Autodesk CAD Manager Tools 4.0.
- 4 Under Autodesk CAD Manager Tools 4.0, click Install.

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

Understand DesignCenter Online Content Types

With DesignCenter Online, content is categorized into folders.

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

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

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

To view online content folders in the Category Listing view

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

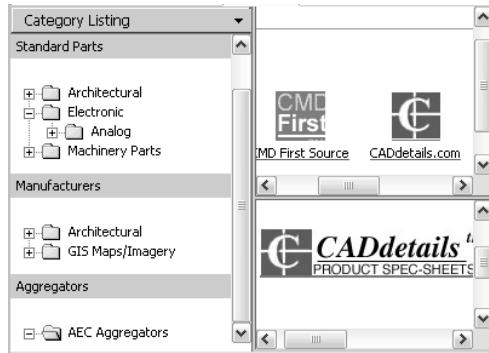
The category folders are displayed in the left pane of the window.

Retrieve Content from the Web

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

Browse for Content

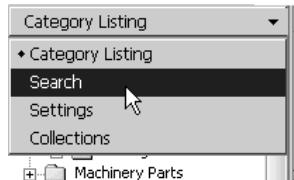
When you use the Category Listing view, you can click the folders in the left pane to view their contents. These folders may contain other folders.



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

Search for Content

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



Control the Number of Categories and Items in a Page

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

Collections

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

Download Content

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

To search for content in DesignCenter Online

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

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

To specify content collections

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

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

To download content to your computer

- 1 In DesignCenter Online, at the top of the left pane, click the heading, and then click Category Listing.
- 2 In the Category folders, click a content item.
- 3 In the Preview area, under the image of the content item, click Save This Symbol As.

- 4** In the Save As dialog box, specify the location on your computer and the file name.
- 5** Click Save.

The content is downloaded to your computer.

To download content to your drawing

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

5

Customize the Drawing Environment

In this chapter

- Set Interface Options
- Create Task-Based Workspaces
- Save and Restore Interface Settings (Profiles)
- Customize Startup

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

Set Interface Options

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

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

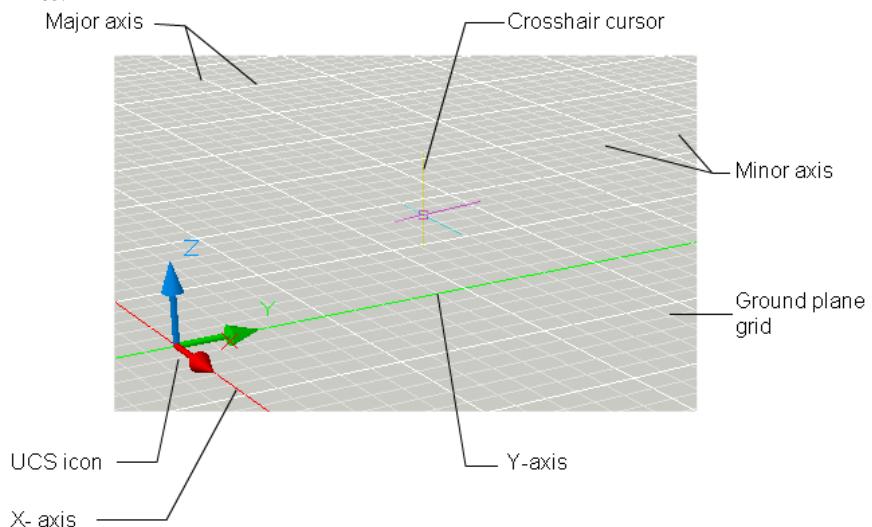
Set Up the Drawing Area

Some settings affect how you work in the drawing area:

- **Background colors (Options dialog box, Display tab, Colors).** You specify the background colors used in model space, layouts, and the block editor. Background colors on the Model tab change to indicate whether you are working in a 2D design context, 3D modeling (parallel projection), or 3D modeling (perspective projection).
- **UCS icon and crosshairs cursor (Options dialog box, 3D Modeling tab).** You specify that the 3D display options and labels for the UCS icon can be set in the 3D Modeling tab of the Options dialog box.
- **Color assignments for X, Y, and Z (Options dialog box, Display tab, Colors).** In 3D views, any interface elements that are associated with the UCS X, Y, and Z axis use special color assignments. The X axis is colored or tinted red, the Y axis is green, and the Z axis is blue. These tints can be turned on or off in the Drawing Window Colors dialog box.
- **Clean Screen.** To expand the drawing display area, click View menu ► Clean Screen to display only the menu bar, status bar, and command window. A check mark is displayed next to the option. Click Clean Screen again to restore the previous setup. A clean screen button is available in the lower-right corner of the application window.
- **View Transitions.** You can control whether view transitions are smooth or instantaneous when you pan, zoom, or change from one view to another (VTOPTIONS command). The default is a smooth transition.
- **Tooltips.** Tooltips provide pop-up information for many drafting aids such as object snaps and dynamic input. The TOOLTIPS system variable turns this information display on and off. The TOOLTIPTMERGE system variable can combine the display of information into one tooltip. You can set the general appearance of tooltips in the Tooltip Appearance Dialog Box

3D Modeling with Perspective Projection

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



Switch Between Model Space and Layouts

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

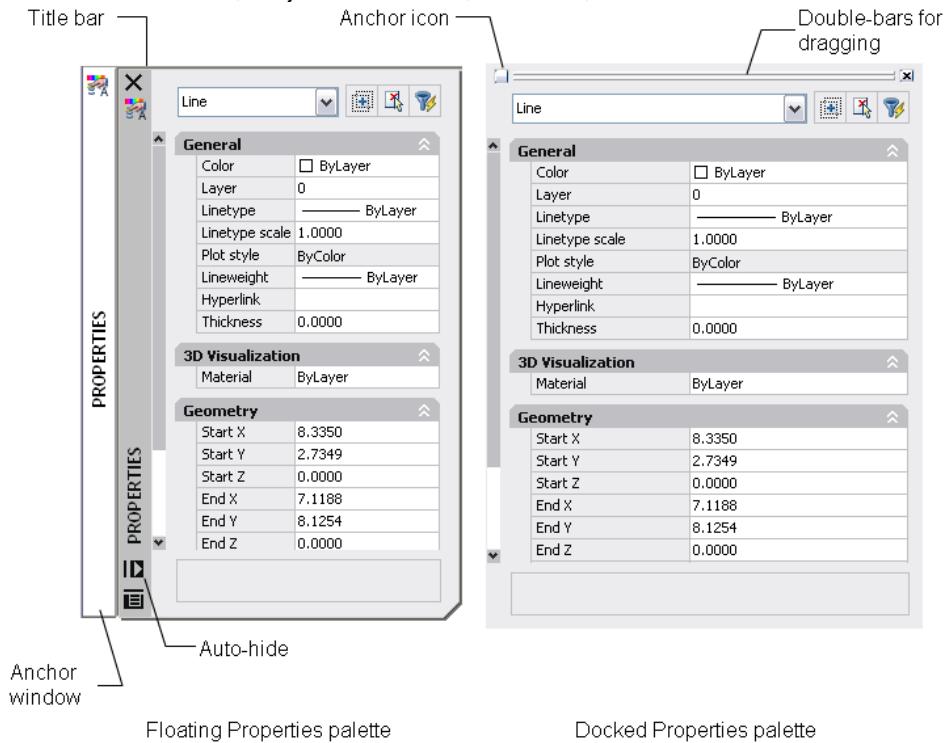
NOTE Access to all shortcut menu options is available from the tabs only.

Specify Application Fonts

- **Font (Options dialog box, Display tab).** Change the fonts used in the application window and in the text window. This setting does not affect the text in your drawings.

Control the Display of Dockable Windows

Many windows, such as the Properties palette, a tool palette, and DesignCenter, are dockable. That is, they can be docked, anchored, or floated.



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

- **Resize.** Drag an edge of the window to change its size. If the window has panes, drag the bar between panes to resize the panes.

- **Allow Docking.** Select this option if you want to dock or anchor a dockable window. A docked window adheres to one side of the application window, causing the drawing area to be resized.
- **Anchor.** Attach, or anchor, a dockable window or palette to the left or right side of the drawing area. An anchored window rolls open and closed as the cursor moves across it. When an anchored window is open, its content overlaps the drawing area. An anchored window cannot be set to stay open. The Allow Docking option must be selected before you can anchor a window.
- **Auto-hide.** A floating window rolls open and closes as the cursor moves across it. When this option is cleared, the window stays open continuously.
- **Transparency.** The window becomes transparent so that it does not obscure objects under it. This option is not available for all windows.

Control the Display of Toolbars

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

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

Lock the Position of Toolbars and Dockable Windows

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

See also:

[Customize the User Interface in the *Customization Guide*](#)

To set options

- 1 Click Tools menu ▶ Options.
- 2 In the Options dialog box, click the tab you want.
- 3 Set options as desired.

- 4** Do either or both of the following:
- Click Apply to record the current options settings in the system registry.
 - Click OK to record the current options settings in the system registry and close the Options dialog box.

Command line: OPTIONS

To customize the colors of the application window elements

- 1** Click Tools menu ► Options.
- 2** In the Options dialog box, Display tab, click Colors.
- 3** In the Drawing Window Colors dialog box, select context and then the interface element you want to change.
- 4** Select the color you want to use from the Color list.
To specify a custom color, select Select Color from the Color list.
- 5** If you want to revert to the default colors, click Restore Current Element, Restore Current Context, or Restore All Contexts.
- 6** Click Apply and Close to record the current option settings in the system registry and close the dialog box.
- 7** Click OK to close the Options dialog box.

Command line: OPTIONS

To change the Model and layout tabs to status bar buttons

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

To turn on the Model and layout tabs

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

To change the font displayed in the command window

- 1** Click Tools menu ► Options.
- 2** In the Options dialog box, Display tab, under Window Elements, click Fonts.

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

Command line: OPTIONS

To lock the position and size of toolbars and dockable windows

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

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

Command line: LOCKUI

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

To anchor a dockable window

- 1 Right-click a floating window or palette title bar. Click Allow Docking.
- 2 Right-click the window or palette title bar. Click Anchor Right or Anchor Left.

To float an anchored window

Do one of the following:

- When the anchored window is open, drag the window title bar away from the anchor tab base.

- Double-click the anchor tab.
- Right-click the window title bar. Clear Allow Docking.

The window is detached from the side of the drawing area.

To change the appearance of an anchor tab base

Do one of the following:

- 1 To represent anchored windows with an icon, right-click the anchor tab base. Click Icons Only.
- 2 To display both an icon and text for anchored windows, right-click the anchor tab base. Click Icons and Text.

To dock a window or palette

- 1 Right-click the window or palette title bar. Verify that Allow Docking is selected.
- 2 Drag the window or palette to a docking location on the right or left side of the drawing area.
- 3 When the outline of the window is displayed in the docking area, release the button.

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

To undock a window or palette

Use one of the following methods:

- Double-click the double bars on the side or top of the window.
- Position the cursor on the double bars at the top or side of the window, hold down the left button on your pointing device, and drag the window away from its docked location.

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

Do one of the following:

- Right-click the title bar of the floating palette or window. Click Auto-hide.
- On the title bar of a floating palette or window, click the Auto-hide button.

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

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

To change the appearance of view transitions

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

Create Task-Based Workspaces

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

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

For more information about using the dashboard, see The Dashboard.

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

- 3D Modeling

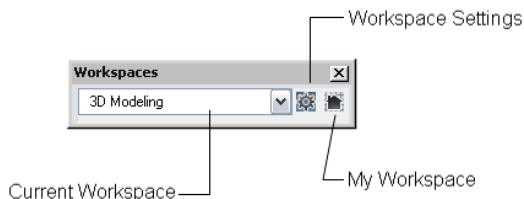
■ AutoCAD Classic

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

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

Switch Workspaces

You can switch to another workspace whenever you need to work on a different task.



Create or Change a Workspace

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

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

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

Select a Sample Workspace

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

You can find the sample workspace in the following location:

C:\Documents and Settings\<user name>\Application Data\Autodesk\AutoCAD 2007\R17.0\<product language>\Support\acadSampleWorkspaces.CUI

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

Workspaces and Profiles

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

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

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

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

To switch workspaces

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

Command line: WORKSPACE

To change workspace settings

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

Command line: WORKSPACE

Command line: WSSETTINGS

To save a workspace

- 1 In the drawing area, arrange a drawing environment by moving, deleting, or displaying those toolbars and dockable windows you want to preserve for future use.

- 2** Click Window menu ► Workspaces ► Save Current As.
- 3** In the Save Workspace dialog box, enter a name for the new workspace or select a name from the drop-down list.
- 4** Click Save.

Command line: WORKSPACE

Command line: WSSAVE

Save and Restore Interface Settings (Profiles)

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

Profiles store settings such as the following:

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

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

Use Profiles for Shared Computers or Different Projects

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

Transfer Profiles between Computers

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

If you make changes to your current profile during a work session and you want to save those changes in an ARG file, you must re-export the profile. When you export the profile with the current profile name, the ARG file is updated with the new settings.

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

To make a profile current

- 1 Click Tools menu ► Options.
- 2 In the Options dialog box, Profiles tab, select the profile you want to make current.
- 3 Click Set Current.
- 4 Click OK.

Command line: OPTIONS

To save a profile

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

Command line: IONS

To make a profile current before starting the program

- 1 On the Windows desktop, right-click the program icon. Click Properties.
 - 2 In the AutoCAD Properties dialog box, Shortcut tab, under Target, enter **/p currentprofile** after the current target directory. For example, to make the profile User12 current, enter the following in Target:
"c:\Program Files\<current release name>\acad.exe"/p user12
 - 3 Click OK.
- The profile name you enter is the current profile each time you start the program.

Customize Startup

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

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

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

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

		of folders you can specify in the path is 15. Each folder name is delimited by semicolons.
/r	Default system pointing device	Restores the default system pointing device. It creates a new configuration file (<i>acad2006.cfg</i>) and renames the previous <i>acad2006.cfg</i> file to <i>acad.bak</i> .
/nologo	No AutoCAD logo screen	Starts the program without first displaying the logo screen.
/p	User-defined registry profile for starting the program	Specifies a user-defined registry profile for starting the program. The selected profile is in effect only for the current session of the program, unless you make another profile current in the Options dialog box during that session. You create or import profiles on the Profiles tab in the Options dialog box. With the /p switch, you can specify only those profiles that are listed in the Options dialog box. If the profile does not exist, the current profile is used.
/nossm	No Sheet Set Manager window	Suppresses the display of the Sheet Set Manager window on startup.
/set	Sheet set	Loads the named sheet set on startup. Use the following format: <path>\<sheet set data file>.DST
/w	Default workspace	Designates which workspace in the loaded CUI files should be restored on startup.
/pl	Background plotting/publishing	Publishes a drawing set descriptions (DSD) file in the background. Use the following format: <path>\<drawing set descriptions file>.DSD

The syntax for using command line switches is

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

When using a switch option, you must follow the switch with a space and then the name of a file, path, or view within quotation marks. For example, the following entry starts the program from a folder named *AutoCAD 2007* with the drawing template *arch1.dwt*, restores a named view *PLAN1*, and executes a script file *startup.scr*.

`"d:\ AutoCAD 2007\acad.exe"/t "d:\AutoCAD 2007\template\arch1" /v "plan1" /b "startup"`

The environment settings are resolved in the following way:

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

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

To start the program with a command line switch

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

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

Valid switches are as follows:

b	/	Script name (<i>b</i> stands for batch process)
t	/	Template file name
c	/	Configuration folder
v	/	View name
s	/	Support folders
r	/	Default system pointing device
ogo l on/		No AutoCAD logo screen
p	/	User-defined registry profile
ms s on/		No Sheet Set Manager window
t e s	/	Sheet Set name
w	/	Default workspace
l p		Background plotting/publishing for a Drawing Set Descriptions (DSD) file

For example, enter "**d:\AutoCAD 2007\ acad.exe** " /**t "d:\AutoCAD 2007\template\arch1"** /**v "plan1"** /**b "startup"**

- 3 Click OK.

6

Tool Palettes

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

In this chapter

- Create and Use Tools from Objects and Images
- Create and Use Command Tools
- Change Tool Palette Settings
- Control Tool Properties
- Customize Tool Palettes
- Organize Tool Palettes
- Save and Share Tool Palettes

Create and Use Tools from Objects and Images

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

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

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

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

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

For block and xref tools, you can choose to be prompted at the command line for a rotation angle (starting from 0) when you click and place the block or xref. When you select this option, the angle that is specified under Rotation in the Tool Properties dialog box is ignored. The prompt for a rotation angle is not shown if you drag the block or xref or, if, at the initial insertion command line prompt, you enter **rotate**.

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

Scale Blocks Automatically

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

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

See also:

- “Control Tool Properties” on page 90
- “Customize Tool Palettes” on page 94
- “Create Task-Based Workspaces” on page 69
- “Add Content with DesignCenter” on page 49

To display the Tool Palettes window

- Click Tools menu ► Palettes ► Tool Palettes. Alternatively, you can press CTRL+3.

Standard toolbar



Command line: TOOLPALETTES

To create a tool from an object in the current drawing

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

The black line indicates where the tool will be located.

- 3 Release the mouse button.

Standard toolbar



Command line: TOOLPAlettes

To use a tool created from a geometric object

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

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

Standard toolbar



Command line: TOOLPAlettes

To use a tool created from a dimension

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

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

Standard toolbar



Command line: TOOLPAlettes

To use a tool on a tool flyout

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

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

Standard toolbar



Command line: TOOLPALETTES

To add or remove a tool flyout

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

NOTE If you remove the flyout from a tool but then add the flyout back, the image, name, and description (the tooltip) that are shown on the tool palette for each tool on the flyout will not be accurate. To correct this, you must return the image, name, and description of the flyout tool to the default settings. See “To change the image, name, and description of a flyout tool to the default settings” on page 93.

Standard toolbar



Command line: TOOLPALETTES

To customize a tool flyout

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

Standard toolbar



Command line: TOOLPALETTES

To use a tool created from a hatch

- 1 On a tool palette, click a hatch tool and drag it to an object in the drawing.
- 2 Release the mouse button to apply the hatch to the object.
The hatch you create will have the same hatch style and properties as the tool you selected from the tool palette.

Standard toolbar



Command line: TOOLPALETTES

To use a tool created from a gradient fill

- 1 On a tool palette, click a gradient fill tool and drag it to an object in the drawing.
- 2 Release the mouse button to apply the gradient fill to the object.
The gradient fill you create will have the same style and properties as the tool you selected from the tool palette.

Standard toolbar



Command line: TOOLPALETTES

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

- 1 On a tool palette, click the block, xref, or raster image you want to insert into your drawing.
- 2 Drag the block, xref, or raster image to the appropriate place in the drawing.
- 3 Release the mouse button to insert the block, xref, or raster image.
The block, xref, or raster image you insert will have the same properties as the tool selected from the tool palette.

Standard toolbar



Command line: TOOLPALETTES

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

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

NOTE When you select this option, the angle that is specified under Rotation in the Tool Properties dialog box is ignored. The prompt for a rotation angle is not shown if you drag the block or xref or, if at the initial insertion command line prompt, you enter **rotate**.

Standard toolbar



Command line: TOOLPALETTES

Create and Use Command Tools

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

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

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

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

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

To create a command tool from a toolbar button

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

If the required toolbar is not displayed, right-click any displayed toolbar and select another toolbar from the list.

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

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

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

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

- 4 Release the mouse button.

- 5 In the Customize dialog box, click Close.

Command line: CUSTOMIZE

To create a command tool from the Customize User Interface

Review final design for any changes LA - 10/17/05

- 1 Right-click on the title bar of a tool palette. Click Customize Commands to open the Customize User Interface (CUI) dialog box.
If the Tool Palettes window is covered up, move the CUI dialog box off to the side.
- 2 In the Command List pane, drag a command to the tool palette and, without releasing the mouse button, move the cursor to the place on the tool palette where you want the tool.
The black horizontal line indicates where the tool will be located.
- 3 Release the mouse button.
- 4 In the CUI dialog box, click Close.

Command line: CUI

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

- 1 Right-click on the title bar of a tool palette. Click Customize Palettes.
- 2 In the program, drag a command from a toolbar to the tool palette and, without releasing the mouse button, move the cursor to the place on the tool palette where you want the tool.
- 3 Release the mouse button.
- 4 On the tool palette, right-click the tool. Click Properties.
- 5 In the Tool Properties dialog box, change the name and description to an appropriate name and description for the string, AutoLISP routine, or script.
- 6 Under Command, in the Command String box, enter a string of commands or customized commands, such as an AutoLISP routine, a VBA macro or application, or a script.
- 7 Click OK.

Command line: CUSTOMIZE

To use a command tool

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

Change Tool Palette Settings

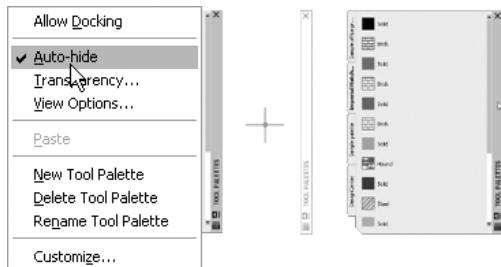
The options and settings for tool palettes are accessible from shortcut menus that are displayed when you right-click in different areas of the Tool Palettes window.

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

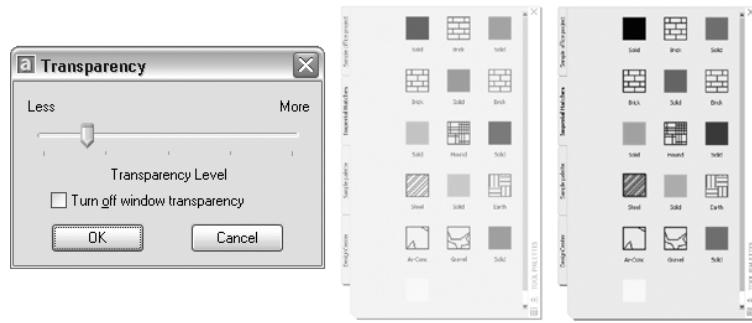
Tool palette settings are saved with your profile.

These settings include

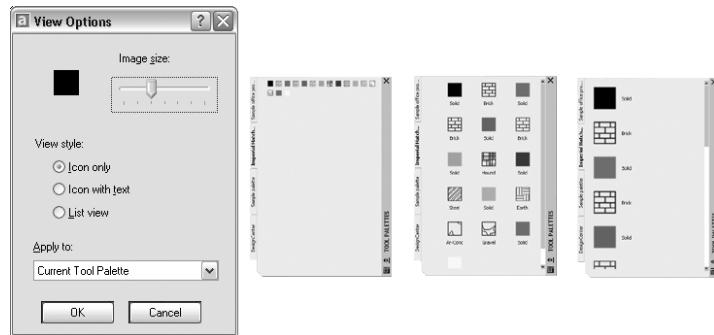
- **Allow Docking.** Toggles the ability to dock or anchor palette windows. If this option is selected, a window can be docked when you drag it over a docking area at the side of a drawing. A docked window adheres to the side of the application window and causes the drawing area to be resized. Selecting this option also makes Anchor Right and Anchor Left available.
- **Anchor Left or Anchor Right.** Attaches the palette to an anchor tab base at the left or right side of the drawing area. The palette rolls open and closed as the cursor moves across it. When an anchored palette is open, its content overlaps the drawing area. An anchored palette cannot be set to stay open.
- **Auto-hide.** Controls the display of the palette when it is floating. When this option is selected, only the tool palette title bar is displayed when the cursor moves outside the tool palette. When this option is cleared, the palette stays open continuously.



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



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



To change the rollover behavior of the Tool Palettes window

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

Rollover behavior is on.



Rollover behavior is off.



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

To change the transparency of the Tool Palettes window

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

- 2** In the Transparency dialog box, adjust the level of transparency for the Tool Palettes window. Click OK.

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

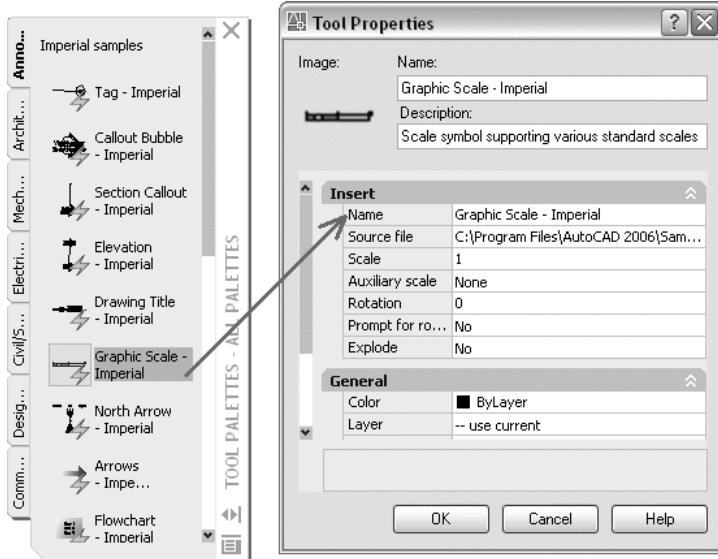
To change the icon display style in the Tool Palettes window

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

Control Tool Properties

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

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



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

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

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

Update the Icon for a Tool

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

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

Specify Overrides for Tool Properties

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

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

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

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

To display the properties of a tool on a tool palette

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

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

- 3 Click OK.

To change the property of a tool on a tool palette

- 1 On a tool palette, right-click a tool. Click Properties.
- 2 In the Tool Properties dialog box, click any property in the list of properties and specify the new value or setting.
 - Properties listed under the Insert or Pattern category control object-specific properties such as scale, rotation, and angle.
 - Properties listed under the General category override the current drawing property settings such as layer, color, and linetype.
 - Auxiliary scale for a block or a hatch tool overrides the regular scale setting when the tool is used. (An auxiliary scale multiplies your current scale setting by the plot scale or the dimension scale.)

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

- 3 Click OK.

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

Standard toolbar



Command line: TOOLPALETTES

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

- 1 On a tool palette, right-click a tool.
- 2 Click Update Tool Image.

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

Standard toolbar



Command line: TOOLPAlettes

To change the image of a tool on a tool palette

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

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

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

NOTE If you specify an image for a tool that has a flyout, that image is displayed on the tool palette for each tool on the flyout. To return the flyout tool images to their default images, see “To change the image, name, and description of a flyout tool to the default settings” on page 93.

Standard toolbar



Command line: TOOLPAlettes

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

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

- 5 Click OK.

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

Standard toolbar



Command line: TOOLPALETTES

Customize Tool Palettes

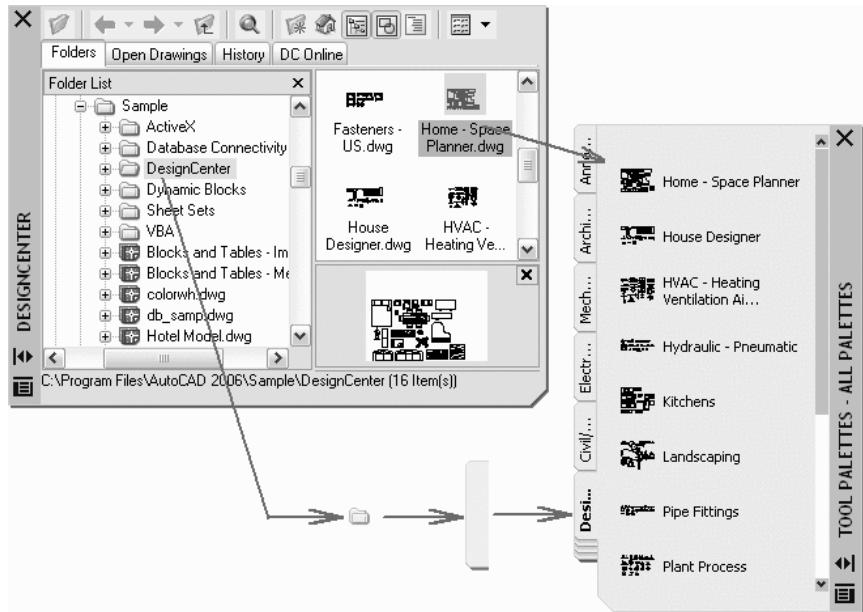
You can add tools to a tool palette with several methods.

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

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

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

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



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

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

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

Read-Only Tool Palettes

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

To apply a read-only attribute to a tool palette, right-click the tool palette (ATC) file in the following location: *C:\documents and settings\<user>*

*name>\application data\autodesk\AutoCAD
2007\r17.0\enu\support\ToolPalette\Palettes.* On the shortcut menu, click Properties. On the General tab, select Read-only, and click OK.

To create a tool palette

- 1 Right-click a blank area inside the Tool Palettes window. Click New Palette.
- 2 In the text box, enter a name for the new palette.
- 3 If necessary, right-click over the tab and select Move Up or Move Down to change the order in which the tab appears in.

Standard toolbar



Command line: TOOLPALETTES

To create a tool palette from a folder or a drawing

- 1 If DesignCenter is not already open, click Tools menu ► Palettes ► DesignCenter.
- 2 In the DesignCenter tree view or the content area, right-click a folder, drawing file, or block. Click Create Tool Palette.
A new tool palette is created that contains all the blocks and hatches in the selected folder or drawing.

Standard toolbar



Command line: ADCENTER

To rename a tool palette

- 1 Right-click a blank area inside the Tool Palettes window. Click Rename Palette.
- 2 In the text box, enter a new name for the palette.

Standard toolbar



Command line: TOOLPALETTES

To delete a tool palette

- 1 Right-click a blank area inside the Tool Palettes window. Click Delete Palette.
- 2 In the Confirm Palette Deletion message box, click OK to delete the tool palette.

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

Standard toolbar



Command line: TOOLPALETTES

To change the position of a tool palette

- 1 Right-click the tab that represents the tool palette you want to move in the Tool Palettes Window. Click Move Up or Move Down.
- 2 Repeat step 1 until the tool palette has been moved to the desired location.

NOTE The order of a tool palette can also be changed using the To change the order of displayed tool palettes within a group.

Standard toolbar



Command line: TOOLPALETTES

To add text to a tool palette

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

- 2** In the text box, add the text you want to display in the window.
- 3** If necessary, drag the text to the appropriate location in the window.

Standard toolbar



Command line: TOOLPAlettes

To add a separator line to a tool palette

- 1** Right-click a blank area inside the Tool Palettes window. Click Add Separator.
- 2** If necessary, drag the separator to the appropriate location in the window.

Standard toolbar



Command line: TOOLPAlettes

To sort items in a tool palette

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

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

Standard toolbar



Command line: TOOLPAlettes

Organize Tool Palettes

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

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

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

To create a tool palette group

- 1 Right-click on the title bar of a tool palette. Click Customize Palettes.
- 2 In the Customize dialog box, Tool Palettes tab, under Palette Groups, right-click on the lower, blank area. Click New Group.
If there are no groups listed in the Palette Groups area, you can create a group by dragging a tool palette from the Tool Palettes area into the Palette Groups area.
- 3 Enter a name for the tool palette group.
- 4 Click Close.

Command line: CUSTOMIZE

To add a tool palette to a tool palette group

- 1 Right-click on the title bar of a tool palette. Click Customize Palettes.
- 2 In the Customize dialog box, Tool Palettes tab, drag a tool palette from the Tool Palettes area into a group in the Palette Groups area.
- 3 Click Close.

Command line: CUSTOMIZE

To remove a tool palette from a tool palette group

- 1 Right-click on the title bar of a tool palette. Click Customize Palettes.
- 2 In the Customize dialog box, Tool Palettes tab, under Palette Groups, right-click the name of the tool palette you want to remove. Click Remove.
You can also drag the tool palette into the Tool Palettes area to remove it from a group.
- 3 Click Close.

Command line: CUSTOMIZE

To display a tool palette group

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

- 2 Click the name of the tool palette group that you want to be shown.

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

Command line: CUSTOMIZE

To delete a tool palette group

- 1 Right-click on the title bar of a tool palette. Click Customize Palettes.
- 2 In the Customize dialog box, Tool Palettes tab, under Palette Groups, right-click the tool palette group you want to delete. Click Delete.

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

- 3 Click Close.

Command line: CUSTOMIZE

To delete all tool palette groups

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

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

- 2 Right-click on the title bar of a tool palette. Click Customize Palettes.
- 3 In the Customize dialog box, Tool Palettes tab, under Palette Groups, right-click a tool palette group. Click Delete.
- 4 Repeat step 3 until all tool palette groups are deleted.
- 5 When you are finished, click Close.

Command line: CUSTOMIZE

To rename a tool palette group

- 1 Right-click on the title bar of a tool palette. Click Customize Palettes.
- 2 In the Customize dialog box, Tool Palettes tab, under Palette Groups, right-click the tool palette group you want to rename. Click Rename.

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

Command line: CUSTOMIZE

To rearrange tool palette groups

- 1** Right-click on the title bar of a tool palette. Click Customize Palettes.
- 2** In the Customize dialog box, Tool Palettes tab, under Palette Groups, click a tool palette group and drag it into the new position.
Any other tool palette groups contained in the group you move will also be moved.
- 3** Click Close.

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

Command line: CUSTOMIZE

To create a tool palette group within another one

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

Command line: CUSTOMIZE

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

- 1** Right-click on the title bar of a tool palette. Click Customize Palettes.
- 2** In the Customize dialog box, Tool Palettes tab, under Palette Groups, select the tool palette that you want to copy.
- 3** Press CTRL while you drag the selected tool palette to another group.
When you release the mouse button, a copy of the tool palette is displayed in the new location.
- 4** Click Close.

Command line: CUSTOMIZE

To change the order of displayed tool palettes within a group

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

Command line: CUSTOMIZE

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

- 1 Right-click on the title bar of a tool palette. Click Customize Palettes.
- 2 In the Customize dialog box, Tool Palettes tab, under Tool Palettes, drag a tool palette to the location you want.
When all tool palettes are displayed, they are shown in the order they appear in this list.
- 3 Click Close.

Command line: CUSTOMIZE

To display all tool palettes

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

Save and Share Tool Palettes

You can save and share a tool palette by exporting it or importing it as a tool palette file. You import and export a tool palette by right-clicking the tool palette in the Customize dialog box, and then click Import or Export. Tool palette files have an *.xtp* file extension.

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

- User-created content tools

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

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

Tool palettes can be used only in the version of AutoCAD in which they were created. For example, you cannot use a tool palette that was created in AutoCAD 2007 in AutoCAD 2005.

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

Tool palette groups are saved in profiles.

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

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

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

See also:

[Save and Restore Interface Settings \(Profiles\)](#)

Part 2

Start, Organize, and Save a Drawing

Chapter 7 Start a Drawing

Chapter 8 Open or Save a Drawing

Chapter 9 Repair, Restore, or Recover Drawing Files

Chapter 10 Maintain Standards in Drawings

7

Start a Drawing

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

In this chapter

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

Start a Drawing from Scratch

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

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

Use the Create New Drawing Dialog Box

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

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

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

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

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

Use the Select Template Dialog Box

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

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

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

Use a Default Drawing Template File

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

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

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

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

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

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

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

Command line: NEW

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

- 1 If necessary, set the STARTUP system variable to 0 and the FILEDIA system variable to 1. On the command line, enter **startup** and **0**; then enter **filedia** and **1**.
- 2 Click Tools menu ➤ Options.
- 3 In the Options dialog box, Files tab, in the list of nodes, click the plus sign (+) next to Template Settings. Click the plus sign (+) next to Drawing Template File Location, and specify a folder path.
- 4 Click the plus sign (+) next to Default Template File Name for QNEW, and specify a drawing template file name.
- 5 Click OK.
- 6 On the Standard toolbar, click Qnew.

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

Standard toolbar



Command line: QNEW

Use a Wizard to Start a Drawing

A setup wizard establishes basic drawing settings step by step.

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

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

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

To begin a new drawing using a wizard

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

Command line: NEW

Use a Template File to Start a Drawing

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

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

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

Create a Drawing Template File

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

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

■ Linetypes

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

Recover the Default Drawing Template File

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

If you specify the 3D Modeling workspace, the default drawing template files are *acad3d.dwt* and *acadiso3d.dwt*.

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

To start a drawing by selecting a template file

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

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

If you want to start a new drawing with no template file, click the arrow next to the Open button. Select one of the “no template” options from the list.

Command line: NEW

To create a drawing template file from an existing drawing

- 1 Click File menu ► Open.
- 2 In the Select File dialog box, select the file you want to use as a template.
- 3 Click OK.
- 4 If you want to delete the existing file contents, click Modify menu ► Erase.
- 5 At the Select Objects prompt, enter **all**, and then select the border and title block (if you want to remove them) and enter **r** (Remove).

- 6** Click File menu ► Save As.
- 7** In the Save Drawing As dialog box, under Files of Type, select the Drawing Template file type.
DWT files must be saved in the current drawing file format. To create a DWT file in a previous format, save the file in the desired DWG format, and then rename the DWG file using a DWT extension.
- 8** In the File Name box, enter a name for the template.
- 9** Click Save.
- 10** Enter a description of the template.
- 11** Click OK.

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

Standard toolbar



Command line: OPEN

To recover the default drawing template files

- 1** Click File menu ► New.
- 2** In the Select Template dialog box, click the arrow next to the Open button. Select one of the options from the list:
 - Open with No Template - Imperial (to restore *acad.dwt*)
 - Open with No Template - Metric (to restore *acadiso.dwt*)The drawing opens with the default settings.
- 3** Click File menu ► Save As.
- 4** In the Save Drawing As dialog box, select the Drawing Template file type. Save the drawing with its original name, either *acad.dwt* for imperial or *acadiso.dwt* for metric.
- 5** Click Save.

NOTE For the 3D Modeling workspace, the default drawing template files are *acad3d.dwt* and *acadiso3d.dwt*

Command line: NEW

Specify Units and Unit Formats

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

Determine the Units of Measurement

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

Convert Drawing Units

If you start a drawing in one system of measurement (imperial or metric) and then want to switch to the other system, use **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 906](#)

To set the units format and precision

- 1 Click Format menu ▶ Units.
- 2 In the Drawing Units dialog box, under Length, select a unit format and precision.
The Sample Output area shows an example of the unit format at the current precision.
- 3 Click OK.

Command line: UNITS

To convert a drawing from inches to centimeters

- 1 Click Modify menu ▶ Scale.
- 2 At the Select Objects prompt, enter **all**.

All objects in the drawing are selected for scaling.

- 3 Enter a base point of ***0,0**.

Scaling will be relative to the world coordinate system origin and the location of the drawing origin will remain at the WCS origin.

- 4 Enter a scale factor of **2.54** (there are 2.54 centimeters per inch).

All objects in the drawing are now 2.54 times larger, corresponding to the equivalent distance in centimeters.

Modify toolbar



Command line: SCALE

To convert a drawing from centimeters to inches

- 1 Click Modify menu ► Scale.

- 2 At the Select Objects prompt, enter **all**.

All objects in the drawing are selected for scaling.

- 3 Enter a base point of ***0,0**.

Scaling is performed relative to the world coordinate system origin, and the location of the drawing origin will remain at the WCS origin.

- 4 Enter a scale factor of 0.3937 (the inverse of 2.54 centimeters per inch).

All objects in the drawing are now smaller, corresponding to the equivalent distance in inches.

Modify toolbar



Command line: SCALE

Set Linear Unit Conventions

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

You can choose from several common conventions to represent the display style and the precision of distances and coordinates. Depending on what you

specify, you can enter and display in decimal format, fractional format, or other notation. These settings affect

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

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

Understand Rounding and Precision

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

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

Enter Distances in Imperial Architectural Format

In architectural format, to enter feet and inches, indicate feet using the prime symbol ('), for example, 7'2". You don't need to enter quotation marks ("") to specify inches.

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

See also:

["Set the Scale for Dimensions" on page 906](#)

To set the units format and precision

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

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

- 3 Click OK.

Command line: UNITS

To convert a drawing from inches to centimeters

- 1 If necessary, click the Model tab to make the Model tab current.
- 2 Click Modify menu ► Scale.
- 3 At the Select Objects prompt, enter **all**.
All objects in the drawing are selected for scaling.
- 4 Enter a base point of ***0,0**.
Scaling will be relative to the world coordinate system origin and the location of the drawing origin will remain at the WCS origin.
- 5 Enter a scale factor of **2.54** (there are 2.54 centimeters per inch).
All objects in the drawing are now 2.54 times larger, corresponding to the equivalent distance in centimeters.

Modify toolbar



Command line: SCALE

To convert a drawing from centimeters to inches

- 1 If necessary, click the Model tab to make the Model tab current.
- 2 Click Modify menu ► Scale.
- 3 At the Select Objects prompt, enter all.
All objects in the drawing are selected for scaling.
- 4 Enter a base point of ***0,0**.
Scaling is performed relative to the world coordinate system origin, and the location of the drawing origin will remain at the WCS origin.
- 5 Enter a scale factor of 0.3937 (the inverse of 2.54 centimeters per inch).
All objects in the drawing are now smaller, corresponding to the equivalent distance in inches.

Modify toolbar



Command line: SCALE

Set Angular Unit Conventions

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

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

Work with Surveyor's Angles

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

@72'8"<n45d20'6"e

To set the angle format and precision

- 1 Click Format menu ► Units.
- 2 In the Drawing Units dialog box, under Angle, select an angle type and precision.
The Sample Output area shows an example of the angle type at the current precision.
- 3 Select Direction to specify an angle direction.
The angle direction controls the point from which angles are measured and the direction in which they're measured. The default starting angle, 0 degrees, is toward 3 o'clock (or east), and positive angle measurement is counterclockwise.
- 4 Select the options you want to use.
- 5 Click OK to close each dialog box.

Command line: UNITS

Add Identifying Information to Drawings

You can keep track of your drawings more easily if you add keywords or other information to them.

Use Windows Explorer

Drawing properties can help you identify a drawing. Some drawing properties are stored by the operating system; for example, drawing type, location, and size. These values are read-only in the drawing file and can only be changed through Windows Explorer.

Use the Find Tool

The Find tool in a standard file selection dialog box can use this information. For example, you can search for all files created on a certain date, or for files you modified yesterday.

Use DesignCenter

You can create additional properties in drawing files. You can store author, title, and subject, and you can assign keywords, hyperlink addresses or directory paths, and custom properties to your drawings. The Advanced tab in the Search tool in DesignCenter can use these properties to locate drawing files. For more information about DesignCenter, see ADCENTER and “Access Content with DesignCenter” on page 46.

Use Sheet Set Manager Properties

With the Sheet Set Manager, you can assign a sheet title, number, and a description to every sheet in a sheet set. For more information about sheet sets, see “Include Information with Sheets and Sheet Sets” on page 259.

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 823.

To display drawing properties for the active drawing

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

Command line: DWGPROPS

To define drawing properties

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

Command line: DWGPROPS

8

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.

In this chapter

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

Open a Drawing

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

To open a drawing, you can

- Use Open on the File menu to display the Select File dialog box.
- Double-click a drawing in Windows Explorer to launch AutoCAD® and open the drawing. If the program is already running, the drawing opens in the current session rather than in a second session.
- Drag a drawing from Windows Explorer into AutoCAD.
If you drop a drawing anywhere outside the drawing area—for example, the command line or the blank space next to the toolbars—the drawing is opened. However, if you drag a single drawing into the drawing area of an open drawing, the new drawing is not opened but inserted as a block reference.
- Use DesignCenter to open drawings
- Use the Sheet Set Manager to locate and open the drawings in a sheet set

Check Drawing Authenticity

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

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

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

Change the Default Drawing Folder

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

See also:

- “Overview of DesignCenter” on page 44
- “Customize Startup” on page 74
- “Work with Sheets in a Sheet Set” on page 239

To open a drawing

- 1 Click File menu ► Open.
- 2 In the Select File dialog box, select one or more files. Click Open.
The icons on the left side of the dialog box provide quick access to commonly used files and file locations. To reorder the icons, drag them to a new location. To add, modify, or remove an icon, right-click the icon to display a shortcut menu.

Standard toolbar



Command line: OPEN

To change the default drawing folder

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

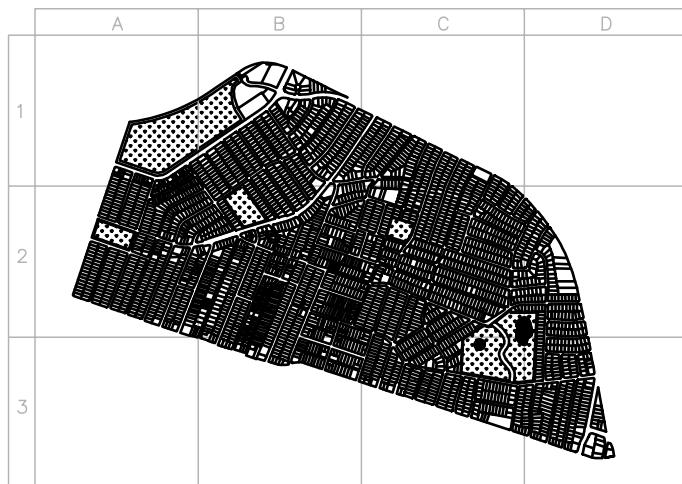
Open Part of a Large Drawing (Partial Load)

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

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

You can only edit what is loaded into the drawing file, but all the drawing's named objects are available in the partially open drawing. Named objects include layers, views, blocks, dimension styles, text styles, viewport configurations, layouts, UCSs, and linetypes.

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



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

To partially open a drawing

- 1 Click File menu ► Open.
- 2 In the Select File dialog box, select a drawing.
- 3 Click the arrow next to Open. Click Partial Open.
- 4 In the Partial Open dialog box, select a view; the default view is EXTENTS.
You can load only geometry from model space views that are saved in the current drawing.

5 Select one or more layers.

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

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

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

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

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

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

8 Click Open.

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

Command line: PARTIALOPEN

To load additional geometry into a partially open drawing

1 Click File menu ► Partial Load.

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

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

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

3 Select one or more layers.

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

- 4 Click Open.

Command line: PARTIALLOAD

Work with Multiple Open Drawings

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

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

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

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

To switch between open drawings

Do one of the following to switch between open drawings:

- Click anywhere in a drawing to make it active.
- Use CTRL+F6 or CTRL+TAB.

- Set the **TASKBAR** command to 1 to display multiple open drawings as separate items on the Windows taskbar. You can then press Alt+TAB to quickly switch from drawing to drawing.

Command line: **TASKBAR**

Save a Drawing

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

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

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

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

Save Part of a Drawing File

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

Save to a Different Type of Drawing File

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

Reduce the Time Required to Save a Drawing File

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

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

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

Work Internationally

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

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

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

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

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

See also:

- “Save Drawings to Previous Drawing File Formats”
- “Export Drawings to Other File Formats”
- “Create Drawing Files for Use as Blocks” on page 464
- “Add Identifying Information to Drawings” on page 119
- “Create and Restore Backup Files” on page 136
- Share Drawing Files Internationally

To save a drawing

- 1 Click File menu ► Save.

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

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

Standard toolbar



Command line: SAVE

To save a drawing automatically

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

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

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

To save selected objects into a new drawing file

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

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

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

Command line: WBLOCK

To save a block definition as a new drawing file

- 1 At the Command prompt, enter **wblock**.
- 2 In the Write Block dialog box, select Block.
- 3 Click in the box next to Block. Select the block definition you want to save.
- 4 In the File Name box, enter the name for the new drawing file.
- 5 In the Location box, specify the folder for the new drawing file.
- 6 In the Insert Units box, you can select a different base unit. This option provides a method to scale objects automatically in the new drawing file.
- 7 Click OK.

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

Command line: WBLOCK

Find a Drawing File

You can search for a drawing using name, location, and date filters, properties such as keywords that you added to the drawing, or text strings containing a specific word or phrase.

- Use the Search tool in Microsoft® Windows® to search for drawings using name, location, and date filters. You can also specify a word or phrase contained in the drawing file. You can search for all textual data except text in tables and fields, and xrefs within drawing files. Supported drawing file types include DWG, DWF, DWT, and DWS.
- Use the Search dialog box for DesignCenter™ to search for Microsoft Windows file properties, such as title or keyword, that you added to drawings.

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

See also:

- "Add Identifying Information to Drawings" on page 119
- "Access Content with DesignCenter" on page 46
- "Work with Sheets in a Sheet Set" on page 239

To search for files

- 1 Click File menu ► Open.
- 2 In the Select File dialog box, click Tools ► Find.
- 3 In the Find dialog box, Name & Location tab, specify a file type, file name, and path.
You can use wild cards when specifying a file name.
- 4 On the Date Modified tab, click All Files, or click Find All Files Created or Modified to specify a date filter.
You can search for drawings modified between a specified range of dates or within a specified number of months or days.
- 5 Click Find Now.
- 6 Select one or more files from the search results. Click OK.
- 7 In the Select File dialog box, click Open.

Standard toolbar



Command line: OPEN

Specify Search Paths and File Locations

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

The Files tab of the Options dialog box is where you set the search path that is used by the program to find drawing support files such as text fonts,

drawings, linetypes, and hatch patterns. The MYDOCUMENTSPREFIX system variable stores the location of the *My Documents* folder for the current user.

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

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

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

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

To change a search path

- 1 Click Tools menu ► Options.
- 2 In the Options dialog box, Files tab, click the plus sign (+) to the left of the kind of path you want to change.
- 3 Select the path you want to change.
- 4 Click Browse, and then search drives and directories until you find the one you want.
- 5 Select the drive and directory you want to use.
- 6 Click OK.

Command line: OPTIONS

9

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.

In this chapter

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

Repair a Damaged Drawing File

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

Repair and Recovery

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

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

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

Example: Auditing Files

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

Auditing Header

DXF NAME Current Value Validation Default

PDMODE 990 - 2040

UCSFOLLOW 811 or 0

Error found in auditing header variables

4 Blocks audited

Pass 1 4 objects audited

Pass 2 4 objects audited

Total errors found 2 fixed 2

If you chose not to correct the errors, the last statement changes to

Total errors found 2 fixed 0.

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

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

To repair a damaged drawing file

- 1 Click File menu ► Drawing Utilities ► Recover.
- 2 In the Select File dialog box, select a file. Click Open.

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

Command line: RECOVER

To repair a damaged drawing file due to a system failure

- 1 If the program encounters a problem and cannot continue, it displays an error message and, for some errors, an error code. Record the error code number, save the changes if possible, and exit to the operating system.
- 2 Restart the program.
- 3 In the Drawing Recovery window, under Backup Files, double-click the drawing node to expand it. On the list, double-click one of the drawing or backup files to open it.

If the program detects that the drawing has been damaged, a message is displayed asking if you want to proceed.

- 4 Enter **y** to proceed.

As the program attempts to repair the drawing, a diagnostic report is displayed. The output from the audit is written to an audit log (ADT) file if the AUDITCTL system variable is set to 1 (On).

- 5 Depending on whether the repair is successful, do one of the following:
 - If the repair is successful, the drawing opens. Save the drawing file.
 - If the program cannot repair the file, a message is displayed. In that case, choose one of the other drawing or backup files listed in the Drawing Recovery window beginning with step 3.

Command line: RECOVER

To repair an open drawing

- 1 Click File menu ► Drawing Utilities ► Audit.
- 2 At the Fix Any Errors Detected? prompt, enter **y** or **n**.

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

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

Command line: AUDIT

To restore a drawing from a backup file

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

Create and Restore Backup Files

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

Use Backup Files

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

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

Save Your Drawing Automatically at Specified Intervals

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

- *Filename* is the current drawing name.

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

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

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

See also:

["Recover from a System Failure" on page 137](#)

To restore a drawing from a backup file

- 1 In Windows Explorer, locate the backup file identified by the *.bak* file extension.
- 2 Select the file you want to rename. You do not need to open it.
- 3 Click File menu ➤ Rename.
- 4 Enter a new name using the *.dwg* file extension.
- 5 Open the file as you would open any other drawing file.

Recover from a System Failure

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

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

Resolve Drawing Files

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

- Drawing files (DWG)
- Drawing template files (DWT)
- Drawing Standards files (DWS)

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

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

- *DrawingFileName_recover.dwg*
- *DrawingFileName_a_b_nnnn.sv\$*
- *DrawingFileName.dwg*
- *DrawingFileName.bak*

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

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

If you close the Drawing Recovery window before resolving all affected drawings, you can open Drawing Recovery at a later time with the DRAWINGRECOVERY command.

Send an Error Report Automatically to Autodesk

If the program encounters a problem and closes unexpectedly, you can send an error report to help Autodesk diagnose problems with the software. The error report includes information about the state of your system at the time the error occurred. You can also add other information, such as what you were doing at the time of the error. The REPORTERROR system variable controls whether the error-reporting feature is available.

To open the Drawing Recovery Manager

- Click File menu ► Drawing Utilities ► Drawing Recovery Manager.
Any drawings that need to be restored from an unexpected program or system failure are listed under Backup Files.

Command line: DRAWINGRECOVERY

To restore a drawing using the Drawing Recovery Manager

- 1 If necessary open the Drawing Recovery Manager.
- 2 In the Drawing Recovery Manager, under Backup Files, double-click a drawing node to list all available drawing and backup files.
- 3 Double-click a file to open it.
If the drawing file is damaged, the drawing is automatically repaired, if possible.

Command line: DRAWINGRECOVERY

To remove a drawing from the Drawing Recovery Manager

- 1 If necessary, click File menu ► Drawing Utilities ► Drawing Recovery Manager.
- 2 Do *one* of the following:
 - Restore the drawing and then save it.
 - Right-click a drawing node. Click Remove.

Command line: DRAWINGRECOVERY

To expand or collapse all nodes in the Drawing Recovery Manager

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

Command line: DRAWINGRECOVERY

To turn error reporting on or off

- 1 At the Command prompt, enter **reporterror**.
- 2 Enter **0** to turn off error reporting, or enter **1** to turn on error reporting.

Maintain Standards in Drawings

10

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

In this chapter

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

Overview of CAD Standards

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

Named Objects for Standards-Checking

You can create standards for the following named objects:

- Layers
- Text styles
- Linetypes
- Dimension styles

Standards File

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

How a Standards Audit Works

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

A standards audit can uncover two types of problems:

- An object with a nonstandard name is present in the drawing being checked. For example, a layer named WALL is present in the drawing but not in any associated standards files.
- A named object in a drawing matches the name of one in a standards file, but their properties are different. For example, in the drawing the WALL layer is yellow, but the standards file specifies red for the WALL layer.

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

Standards Plug-Ins

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

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

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

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

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

Standards Settings

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

Define Standards

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

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

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

To create a standards file

- 1 Click File menu ► New.
- 2 Enter a template file name or press ENTER to continue.
- 3 In a new drawing, create any layers, dimension styles, linetypes, and text styles that you want to be part of the standards file.
- 4 Click File menu ► Save As.
- 5 In File Name, enter a name for the standards file.
- 6 In the Files of Type list, select AutoCAD Drawing Standards (*.dws).
DWS files must be saved in the current drawing file format. To create a DWS file in a previous drawing file format, save the file in the desired DWG format, and then rename the DWG file using a *.dws* extension.
- 7 Click Save.

To associate a standards file with the current drawing

- 1 At the Command prompt, enter **standards**.
- 2 In the Configure Standards dialog box, Standards tab, click the plus (+) button (Add Standards File).
- 3 In the Select Standards File dialog box, locate and select a standards file.

- 4 Click Open.
- 5 (Optional) Repeat steps 2 and 3 if you want to associate additional standards files with the current drawing.
- 6 Click OK.

CAD Standards toolbar



To remove a standards file from the current drawing

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

CAD Standards toolbar



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

- 1 At the Command prompt, enter **standards**.
- 2 In the Configure Standards dialog box, Standards tab, select the standards file whose position you want to change in Standards Files Associated with the Current Drawing.
- 3 Do one of the following:
 - Click the up arrow button (Move Up) to move the standards file up one position in the list.
 - Click the down arrow button (Move Down) to move the standards file down one position in the list.
- 4 (Optional) Repeat steps 2 and 3 if you want to change the position of other standards files in the list.

5 Click OK.

CAD Standards toolbar



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

- 1** At the Command prompt, enter **standards**.
- 2** In the Configure Standards dialog box, Plug-ins tab, do one of the following:
 - Select the check box for at least one plug-in to audit a drawing for standards violations.
 - To select all plug-ins, right-click in the Plug-ins list. Click Select All. (You can clear all plug-ins. Right-click in the Plug-ins list. Click Clear All.)
- 3** Click OK.

CAD Standards toolbar



Check Drawings for Standards Violations

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

After you associate a standards file with a drawing, you should periodically check the drawing to make sure it conforms with its standards. This is especially important when more than one person is updating the drawing file. For example, in a project with multiple subcontractors, one subcontractor may create new layers that do not comply with the standards you have defined. In that case, you need to be able to identify the nonstandard layers and fix them.

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

Check a Single Drawing

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

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

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

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

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

Check Multiple Drawings

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

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

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

Use Notification of Standards Violations While Working in a Drawing

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

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

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

Display Alerts for Named Objects

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

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

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

NOTE If you select Display Alert upon Standards Violation in the Check Standards dialog box and you fix a violation, you can return to a fixing operation where you

left off. If you select Display Standards Status Bar icon and you click the icon and fix a violation, you must return to a standards-fixing operation from the beginning.

To audit a drawing for standards violations

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

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

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

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

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

- 3 Do one of the following:

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

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

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

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

- Click the Next button to display the next violation.

- 4 Repeat steps 2 and 3 until you have cycled through all standards violations.

- 5 Click Close.

CAD Standards toolbar



To turn the display of ignored problems on or off

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

CAD Standards toolbar



To turn notification for standards violations on or off

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

CAD Standards toolbar



To start the Batch Standards Checker

- Click Start menu (Windows) ▶ All Programs ▶ Autodesk ▶ AutoCAD 2007 ▶ Batch Standards Checker.

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

To create a standards check file for the Batch Standards Checker

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

To open an existing standards check file

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

To specify a standards override for a standards check file

- 1 Start the Batch Standards Checker.
- 2 Create a standards check file or open an existing one.
- 3 In the Batch Standards Checker, Standards tab, select the Check All Drawings Using the Following Standards Files option.
- 4 Click the + button (Add Standards File).
- 5 In the Open dialog box, select a standards file to use as an override. Click Open.

- 6** (Optional) Repeat steps 5 and 6 if you want to add additional standards files as overrides.
- 7** In the Batch Standards Checker, click File menu ► Save Check File. (You can also click Save on the Batch Standards Checker toolbar.)

To audit a set of drawings for standards violations

- 1** Start the Batch Standards Checker.
- 2** Open an existing standards check file, or create a new one.
- 3** In the Batch Standards Checker, click Check menu ► Start Check. You can also click Start Check on the Batch Standards Checker toolbar.)
When the batch audit is complete, the report is displayed in a browser window.

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

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

To add notes to the batch audit report

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

To view a previously generated batch audit report

- 1** Start the Batch Standards Checker.
- 2** Open the standards check file that generated the report you want to view.
- 3** In the Batch Standards Checker, click Check menu ► View Report. (You can also click View Report on the Batch Standards Checker toolbar.)
The report is displayed in a browser window.
- 4** Select one of the following options to filter the data displayed in the report:
 - **Overview.** Summarizes the number of problems encountered in each audited drawing.

- **Plug-ins.** Summarizes the standards plug-ins installed on the system used to run the batch audit.
 - **Standards.** Summarizes the standards files used to check the drawings in the batch audit.
 - **Problems.** Provides detailed information about each problem that was encountered.
 - **Ignored Problems.** Provides detailed information about problems that were flagged as ignored.
 - **All.** Shows all available standards audit information.
- 5 In the For list, select individual drawings to view their audit information.

To export a batch audit report

- 1 Start the Batch Standards Checker.
- 2 Create a standards check file or open an existing one.
- 3 In the Batch Standards Checker, click Check menu ▶ Export Report. (You can also click Export Report on the Batch Standards Checker toolbar.)

Translate Layer Names and Properties

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

Convert Layers to Established Drawing Standards

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

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

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

To convert a drawing's layers to standard layer settings

- 1 Click Tools menu ► CAD Standards ► Layer Translator.
- 2 In the Layer Translator, do one of the following:
 - Click Load to load layers from a drawing, a drawing template, or a drawing standards file. In the Select Drawing File dialog box, select the file you want. Click Open.
 - Click New to define a new layer. In the New Layer dialog box, enter a name for the new layer and select its properties. Click OK.
- You can repeat step 2 as many times as you want. If you load other files containing layers of the same name as ones already shown in the Translate To list, the layer properties first loaded in the list are maintained; any duplicate layer properties are ignored.
- 3 Map layers in the current drawing to the layers you want to convert to. Use either or both of the following methods to map layers:
 - To map all identically named layers from one list to the other, click Map Same.
 - To map individual layers, in the Translate From list, select one or more layers. In the Translate To list, select the layer whose properties you want to use. Click Map to define the mapping. You can repeat this method for each layer or group of layers to be translated.

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

- 4 (Optional) You can perform the following tasks in the Layer Translator:
 - To change the properties of a mapped layer, in the Layer Translation Mappings list, select the mapping whose properties you want to change. Click Edit. In the Edit Layer dialog box, change the linetype, color, linewidth, or plot style of the mapped layer. Click OK.
 - To customize the process of layer translation, click Settings. In the Settings dialog box, select the options you want. Click OK.
 - To save layer mappings to a file, click Save. In the Save Layer Mappings dialog box, enter a file name. Click OK.
- 5 Click Translate to perform the layer translations you have specified.

CAD Standards toolbar



Command line: LAYTRANS

View Selected Drawing Layers

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

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

To specify which layers are displayed in the drawing area

- 1 Click Tools menu ▶ CAD Standards ▶ Layer Translator.
- 2 In the Layer Translator, click Settings.
- 3 In the Settings dialog box, do one of the following:
 - To display objects on selected layers in your drawing, select Show Layer Contents When Selected. Only the layers selected in the Layer Translator dialog box are displayed in the drawing area.
 - To display objects on all layers in your drawing, clear Show Layer Contents When Selected.
- 4 Click OK.

CAD Standards toolbar



Command line: LAYTRANS

Purge Unreferenced Layers

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

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

To purge all unreferenced layers from a drawing

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

CAD Standards toolbar



Command line: LAYTRANS

Part 3

Control the Drawing Views

Chapter 11 Change Views

Chapter 12 Use 3D Viewing Tools

Chapter 13 Display Multiple Views in Model Space

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.

In this chapter

- Pan or Zoom a View
- Pan and Zoom with the Aerial View Window
- Save and Restore Views
- Control the 3D Projection Style
- Shade a Model and Use Edge Effects

Pan or Zoom a View

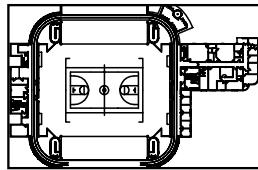
You can pan to reposition the view in the drawing area or zoom to change magnification.

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

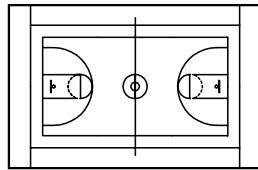
You can change the magnification of a view by zooming in and out, which is similar to zooming in and out with a camera. ZOOM does not change the absolute size of objects in the drawing; it changes only the magnification of the view.

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

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



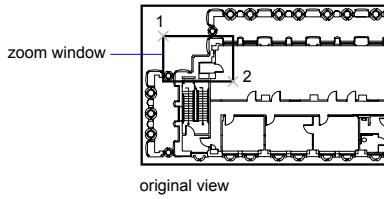
zoomed out



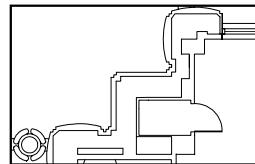
zoomed in

Zoom to Magnify a Specified Rectangular Area

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



original view



new view

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

Zoom in Real Time

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

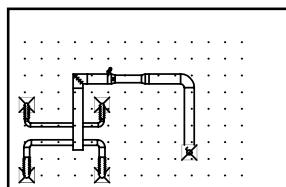
Zoom to Magnify One or More Objects

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

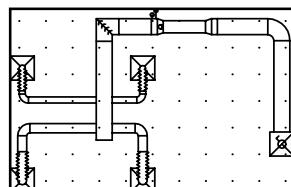
Zoom to View All Objects in the Drawing

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

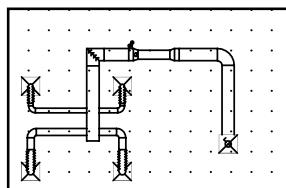
ZOOM All displays either the user-defined grid limits or the drawing extents, whichever view is larger.



current view



zoomed to extents



zoomed to show entire drawing
(all)

See also:

["Pointing Device Buttons"](#)

["Scale Views in Layout Viewports" on page 223](#)

["Toolbars" on page 24](#)

To pan by dragging

- 1 Click View menu ► Pan ► Realtime.
- 2 When the hand cursor appears, hold down the button on your pointing device as you move.
If you are using a wheel mouse, hold down the wheel button and move the mouse.
- 3 Press ENTER, press ESC, or right-click to exit.

Standard toolbar



Command line: PAN

To pan by specifying points

- 1 Click View menu ► Pan ► Point.
- 2 Specify a base point. This is the point you want to change.
- 3 Specify a second (pan to) point. This is the new location for the point you selected first.

Command line: PAN

To zoom by dragging

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

Standard toolbar



Command line: ZOOM

To zoom in to an area by specifying its boundaries

- 1 Click View menu ► Zoom ► Window.
- 2 Specify one corner of the rectangular area you want to view.
- 3 Specify the opposite corner.
You automatically choose the Window option when you specify a point immediately after starting the ZOOM command.

Standard toolbar



Command line: ZOOM

Shortcut menu: With the ZOOM command active, right-click in the drawing area. Click Zoom Window.

To display the drawing extents by zooming

- Click View menu ► Zoom ► Extents.

All objects in the drawing are displayed to be as large as possible and still fit in the current viewport or the drawing area.

Standard toolbar



Command line: ZOOM

Shortcut menu: With the ZOOM command active, right-click in the drawing area. Click Zoom Extents.

To display the area of the grid limits by zooming

- Click View menu ► Zoom ► All.

The limits of the drawing grid fill the current viewport or the drawing area. If there are any objects outside the grid limits, they are also included.

Standard toolbar



Command line: ZOOM

To restore the previous view

- Click View menu ► Zoom ► Previous.

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

Standard toolbar



Command line: ZOOM

Shortcut menu: With the ZOOM command active, right-click in the drawing area. Click Zoom Original.

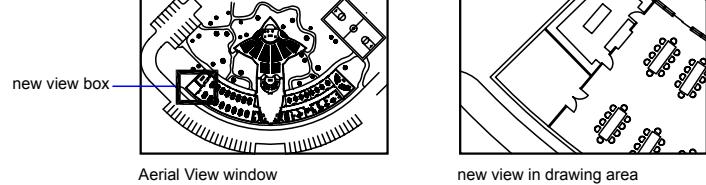
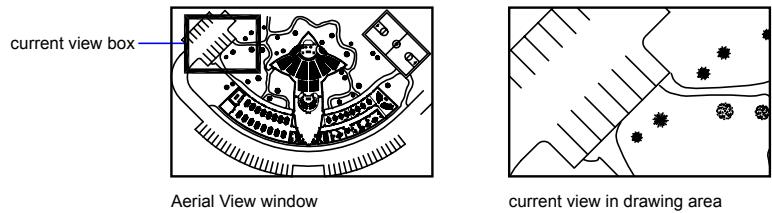
Pan and Zoom with the Aerial View Window

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

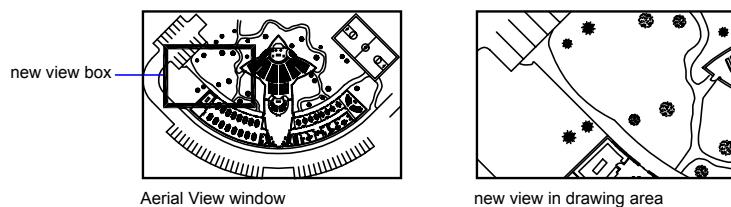
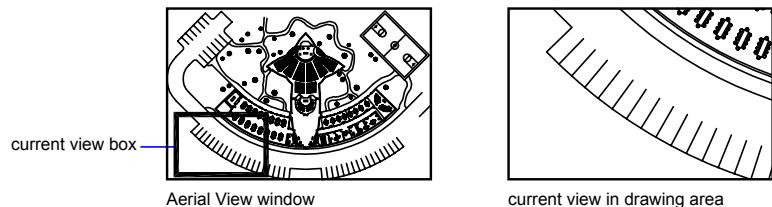
You can use the Aerial View window to change the view in your current viewport quickly. If you keep the Aerial View window open as you work, you can zoom and pan without interrupting your current command. You can also specify a new view without having to choose a menu option or enter a command.

Use the View Box to Pan and Zoom

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



To pan the drawing, move the view box.



Change the View Displayed Inside the Aerial View Window

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

Use the Aerial View Window with Multiple Viewports

Only the view in the current viewport is used in the Aerial View window.

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

To zoom to a new area using the Aerial View window

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

Command line: DSVIEWER

To pan using the Aerial View window

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

Command line: DSVIEWER

To display the entire drawing in the Aerial View window

- 1** Click View menu ► Aerial View.
- 2** In the Aerial View window, click View menu ► Global.

Command line: DSVIEWER

To increase or decrease magnification of the Aerial View image

- 1 Click View menu ► Aerial View.
- 2 On the Aerial View toolbar, click Zoom Out or Zoom In.

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

Command line: DSVIEWER

To turn dynamic updating on and off

- In the Aerial View window, click Options menu ► Dynamic Update.
A check mark indicates that the Aerial View window shows changes as they occur.

To turn viewport updating on and off

- In the Aerial View window, click Options menu ► Auto Viewport.
A check mark indicates that Aerial View displays the current viewport as you switch viewports.

Save and Restore Views

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

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

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

Save a View

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

- Magnification, center point, and view direction
- View category that you assign to the view (optional)

- The location of the view (the Model tab or a specific layout tab)
- Layer visibility in the drawing at the time the view is saved
- User coordinate system
- 3D perspective
- Live section
- Visual style
- Background

Restore a Named View

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

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

To display the previous view

- On the Standard toolbar, click Zoom Previous.

Command line: ZOOM

To display the previous view during PAN Realtime and ZOOM Realtime

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

To save and name a view

- 1 Do one of the following:
 - If you have more than one viewport in model space, click inside the viewport that contains the view you want to save.
 - If you are working in a layout, select the viewport.
- 2 Click View menu ► Named Views.
- 3 In the View Manager, click New.

- 4 In the New View dialog box, View Name box, enter a name for the view. If the drawing is part of a sheet set, the view categories for the sheet set are listed. You can add a category or select one from the list.
- 5 In the Boundary section, select one of the following options to define the area of the view:
 - **Current Display.** Includes all of the drawing that is currently visible.
 - **Define Window.** Saves part of the current display. The dialog box closes while you use the pointing device in the drawing to specify opposite corners of the view. To redefine the window, click the Define View Window button.
- 6 Click OK twice to save the new view.

Command line: VIEW

3D Navigation panel, Previous View

To restore a saved view

- 1 Do one of the following:
 - If you have more than one viewport in model space, click inside the viewport that contains the view you want to restore.
 - If you are working in a layout, select the viewport.
- 2 Click View menu ► Named Views.
- 3 In the View Manager, Views list, select the view that you want to restore.
- 4 Click Set Current.
- 5 Click OK.

View toolbar



Command line: VIEW

3D Navigation panel, Previous View

To rename a view

- 1 Click View menu ► Named Views.

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

View toolbar



Command line: VIEW

3D Navigation panel, Previous View

To change the properties of a view

- 1** Click View menu ► Named Views.
- 2** In the View Manager, click the view name you want to change. If the view isn't already displayed, expand the appropriate View list, and then click a view name.
- 3** In the Properties panel, click the property that you want to change.
- 4** Specify the new property value by entering a new value or by selecting from a list of values.
- 5** Click OK.

View toolbar



Command line: VIEW

3D Navigation panel, Previous View

To delete a named view

- 1** Click View menu ► Named Views.
- 2** In the View Manager, click a view, and then click Delete.
- 3** Click OK.

View toolbar



Command line: VIEW

3D Navigation panel, Previous View

To view a list of saved viewport arrangements

- Click View menu > Viewports > Named Viewports.

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

Viewports toolbar



Command line: VPORTS

Control the 3D Projection Style

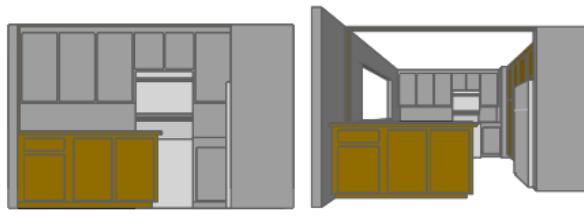
You can view both parallel and perspective projection of a 3D model.

Overview of Parallel and Perspective Views

You can create realistic visual effects in a drawing by defining either parallel or perspective projections of a model.

The difference between perspective views and parallel projections is that perspective views require a distance between a theoretical camera and target point. Small distances produce severe perspective effects; large distances produce mild effects.

The following illustration shows the same model in both a parallel projection and perspective projection. Both are based on the same viewing direction.



Parallel projection

Perspective projection

No procedures for this topic.

Define a Perspective Projection (DVIEW)

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

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

To define a perspective view of a 3D model using DVIEW

- 1 On the command line, enter **dview**.
- 2 Select the objects to display.
- 3 Enter **ca** (Camera).

By default, a camera point is set at the center of the drawing.

- 4 Adjust the view as if you're aiming a camera.

A representation of a house shows the current viewing angle. You can set your view dynamically by moving the crosshairs and clicking.

- 5 To switch between angle-input methods, enter **t** (Toggle Angle).

You also can adjust the view with one of two angle-input methods.

- For the Enter Angle from the XY Plane option, enter the angle of the camera up or down relative to the XY plane of the current UCS. The default setting, 90 degrees, points the camera straight down from above.

After you enter the angle, the camera is locked at that height, and you can rotate the camera about the target with the rotation angle measured relative to the X axis of the current UCS.

- For the Enter Angle in the XY Plane from the X Axis option, rotate the camera about the target with the rotation angle measured relative to the X axis of the current UCS.

- 6 To turn on the perspective view, enter **d** (Distance).
- 7 Specify a distance, or press ENTER to set the perspective view.

You can use the slider bar to set the distance between the selected objects and the camera, or you can enter a real number. If the target and camera points are very close (or if the Zoom option is set high), you might see only a small part of your drawing.

Command line: DVIEW

To turn off a perspective view using DVIEW

- 1 On the command line, enter **dview**.
- 2 Select the objects to display.
- 3 Enter **o** (Off).

Perspective is turned off and the view is restored to a parallel projection.

Command line: DVIEW

Define a Parallel Projection

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

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

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

To create a parallel projection dynamically

- 1 On the command line, enter **dview**.
- 2 Do *one* of the following:
 - Select the objects to display and press ENTER.

- Press ENTER without selecting objects to see a representation of a house that shows the current viewing angle.

3 Enter **ca** (Camera).

By default, a camera point is set at the center of the drawing.

4 Adjust the view as if you're aiming a camera.

You can set your view dynamically by moving the crosshairs and clicking.

5 To switch between the angle-input methods, enter **t** (Toggle Angle).

You can also adjust the view with one of two angle-input methods.

- For the Enter Angle from the XY Plane option, enter the angle of the camera up or down relative to the XY plane of the current UCS. The default setting, 90 degrees, points the camera straight down from above.

After you enter the angle, the camera is locked at that height, and you can rotate the camera about the target with the rotation angle measured relative to the X axis of the current UCS.

- For the Enter Angle in the XY Plane from the X Axis option, rotate the camera about the target with the rotation angle measured relative to the X axis of the current UCS.

6 When you finish specifying the parallel projection, press ENTER.

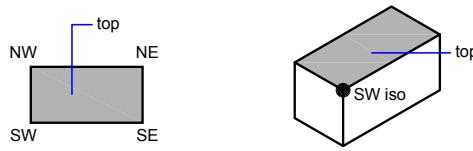
Command line: DVIEW

Choose Preset 3D Views

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

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

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



To use a preset 3D view

- Click View menu > 3D Views. Select a preset view (Top, Bottom, Left, and so on).

View toolbar



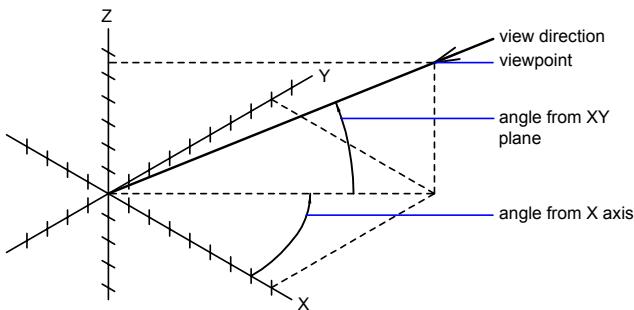
Command line: VIEW

Define a 3D View with Coordinate Values or Angles

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

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

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



To set a view with a viewpoint coordinate

- 1 Click View menu > 3D Views > Viewpoint.
- 2 Click inside the compass to specify the viewpoint. The selected viewpoint is used to view the drawing in the direction of 0,0,0.

Command line: VPOINT

To set a view with two angles of rotation

- 1 At the Command prompt, enter **vpoint**.
- 2 Enter r (Rotate) to specify a new direction using two angles.
- 3 Enter an angle in the XY plane measured from the positive X axis.
- 4 Enter an angle from the XY plane that represents your position while viewing the model in the direction of 0,0,0.

Command line: VPOINT

To set standard views with VPOINT (AEC convention)

- 1 On the command line, enter **vpoint**.
- 2 Enter a coordinate according to the viewpoint you want:
 - Enter **0,0,1** for a top (plan) view.
 - Enter **0,-1,0** for a front view.
 - Enter **1,0,0** for a right side view.

- Enter **1,-1,1** for an isometric view.

Command line: VPOINT

To set standard views with VPOINT (mechanical design convention)

- 1 On the command line, enter **vpoint**.
- 2 Enter a coordinate according to the viewpoint you want:
 - Enter **0,1,0** for a top view.
 - Enter **0,0,1** for a front view.
 - Enter **1,0,0** for a right side view.
 - Enter **1,1,1** for an isometric view. This view is identical to one with a right-out of 45 degrees and a top-out of 35.267 degrees.

Command line: VPOINT

Change to a View of the XY Plane

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

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

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

To change the current view to the XY plane

- 1 Click View menu ► 3D Views. Click Plan View.
- 2 Select one of the following options:
 - Current (for the current UCS)
 - World (for the WCS)
 - Named (for a saved UCS)

NOTE PLAN changes the viewing direction and turns off perspective and clipping; it does not change the current UCS. Any coordinates entered or

displayed subsequent to the PLAN command remain relative to the current UCS.

Command line: PLAN

Shade a Model and Use Edge Effects

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

Use a Visual Style to Display Your Model

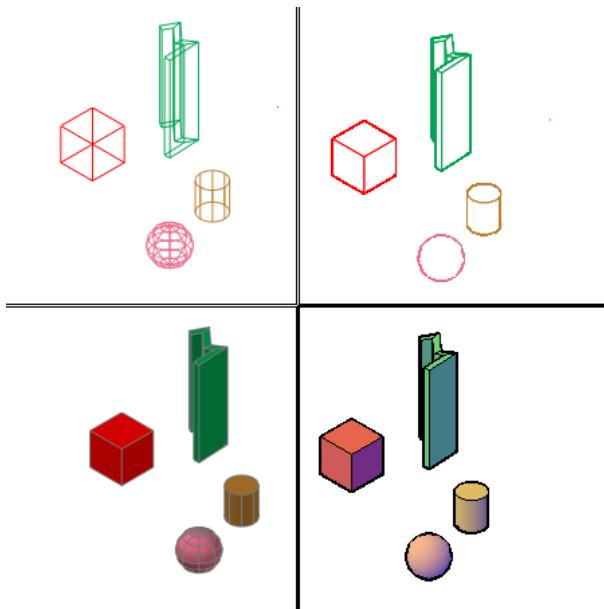
A visual style is a collection of settings that control the display of edges and shading in the viewport. Instead of using commands and setting system variables, you change the properties of the visual style. As soon as you apply a visual style or change its settings, you can see the effect in the viewport.

The Visual Styles Manager displays sample images of the visual styles available in the drawing. The selected visual style is indicated by a yellow border, and its settings are displayed in the panel below the sample images.

When the Dashboard is displayed, you can change some frequently used settings directly or open the Visual Styles Manager.

Five default visual styles are supplied with the product:

- 2D Wireframe. Displays the objects using lines and curves to represent the boundaries. Raster and OLE objects, linetypes, and linewidths are visible.
- 3D Wireframe (upper left in the illustration). Displays the objects using lines and curves to represent the boundaries.
- 3D Hidden (upper right). Displays the objects using 3D wireframe representation and hides lines representing back faces.
- Realistic (lower left). Shades the objects and smooths the edges between polygon faces. Materials that you have attached to the objects are displayed.
- Conceptual (lower right). Shades the objects and smooths the edges between polygon faces. Shading uses the Gooch face style, a transition between cool and warm colors rather than dark to light. The effect is less realistic, but it can make the details of the model easier to see.



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

You can select a visual style and change its settings at any time. The changes you make are reflected in the viewports where the visual style is applied. For more information about face settings, environment settings, and edge settings, see *Customize a Visual Style*. Any changes you make to a visual style are saved in the drawing.

To save a visual style in the drawing

- Enter **vssave** at the Command prompt.

To apply a visual style to a viewport

- 1 Click Tools menu > Palettes > Visual Styles.
- 2 Click in the viewport to make it current.
- 3 In the Visual Styles Manager, double-click the sample image of the visual style.

The selected visual style is applied to the model in the viewport.

An icon in the sample image indicates that the visual style is in use in the current viewport. When you change viewports, the icon changes to indicate that the visual style is in use in the current drawing.

Visual Styles toolbar:



Shortcut menu: Right-click the visual style in the Visual Styles Manager. Click Apply to Current Viewport.

Command line: VISUALSTYLES

Visual Style panel, Visual Styles Manager

To store a visual style on a tool palette

- 1 Click Tools menu ► Palettes ► Visual Styles.
- 2 Click Tools menu ► Palettes ► Tool Palettes.
- 3 In the Tool Palettes window, click the Visual Styles tab.
- 4 In the Visual Styles Manager, select the sample image of the visual style.
- 5 Below the images, click the Export the Selected Visual Style to the Tool Palette button.

Visual Styles toolbar:



Shortcut menu: Right-click the visual style in the Visual Styles Manager. Click Export to Active Tool Palette.

Command line: VISUALSTYLES

Visual Style panel, Visual Styles Manager

To use a visual style from a different drawing

- 1 Open the drawing that has the visual style that you want to use.
- 2 Click Tools menu ► Palettes ► Visual Styles.
- 3 Click Tools menu ► Palettes ► Tool Palettes.
- 4 In the Tool Palettes window, click the Visual Styles tab.
- 5 In the Visual Styles Manager, select the sample image of the visual style.

- 6** Below the images, click the Export the Selected Visual Style to the Tool Palette button.
- 7** Open the drawing where you want to use the visual style.
- 8** On the tool palette, select the visual style.
- 9** Right-click and click Add to Current Drawing.

Visual Styles toolbar:



Shortcut menu: Right-click the visual style in the Visual Styles Manager. Click Export to Active Tool Palette.

Command line: VISUALSTYLES

Visual Style panel, Visual Styles Manager

To restore the original settings of a default visual style

- 1** Click Tools menu ► Palettes ► Visual Styles.
- 2** Select a default visual style.
- 3** Right-click and click Reset to Default.

Visual Styles toolbar:



Command line: VISUALSTYLES

Visual Style panel, Visual Styles Manager

Use 3D 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.

I 2

In this chapter

- Define a 3D View with a Camera
- Create Motion Path Animations

Define a 3D View with a Camera

Define a 3D view by placing a camera in model space and adjusting camera settings to suit your needs.

Overview of Cameras

You can place a camera in a drawing to define a 3D view.

- **Location.** Defines the point from which you are viewing a 3D model.
- **Target.** Defines the point you are viewing by specifying the coordinate at the center of the view.
- **Lens length.** Defines the magnification properties of a camera's lens. The greater the lens length, the narrower the field of view.
- **Front and back clipping planes.** Specifies the location of clipping planes. Clipping planes are boundaries that define, or clip, a view. In the camera's view, everything between the camera and the front clipping plane is hidden. Likewise, everything between the back clipping plane and the target is hidden.

By default, saved cameras are given names such as Camera1, Camera2, and so on. You can rename a camera to better describe its view. The View Manager lists existing cameras in a drawing as well as other named views.

Use the Camera Glyph Appearance Dialog Box to control the camera glyph's colors and size.

No procedures for this topic.

Create a Camera

Set a camera and target location to create and save 3D perspective views of objects.

To create a camera

- 1 Click View menu > Create Camera.
- 2 Click in the drawing to specify the camera's location.
- 3 Do one of the following:
 - If you are finished setting up the camera, press ENTER.

- To further define camera properties, right-click and select from the list of options. Then press ENTER to finish setting up the camera.

View toolbar



Command line: CAMERA

3D Navigate panel, Create Camera

To create a camera from the Tool palette

- 1 Click Tools menu ► Palettes ► Tool Palettes.
- 2 Click the Camera tool palette tab to make it active.

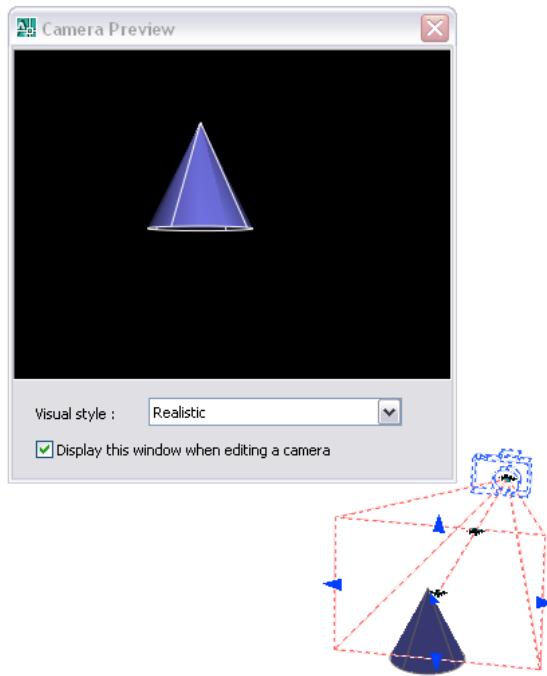
Command line: CAMERA

To display a camera

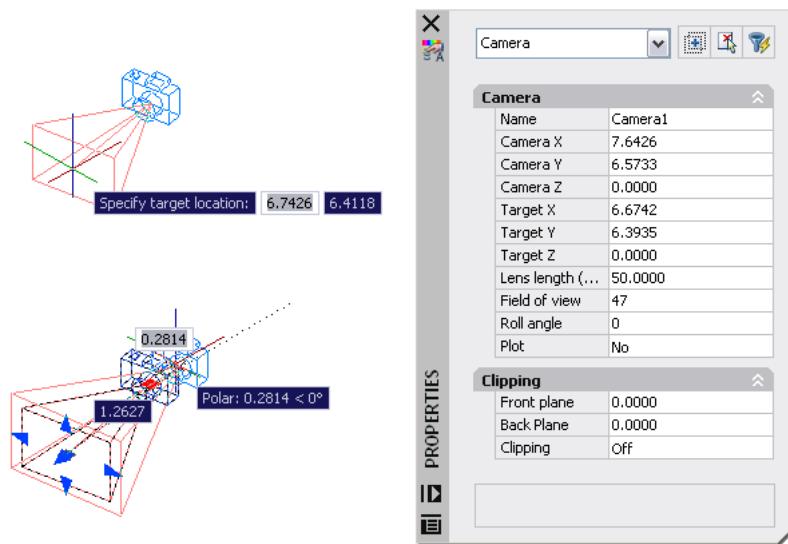
Change Camera Properties

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

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



- Click and drag grips to resize or relocate a lens length or field of view.
- Use the Dynamic Input tooltip to enter X, Y, Z coordinate values.



To change a camera's lens length

- 1 If cameras are not already displayed in the drawing, click View menu ▶ Display ▶ Cameras.
- 2 Click the camera glyph.
- 3 Click a Lens Length/FOV grip tool.
- 4 Move the cursor and click where you want to position the lens.
- 5 Press ENTER.

NOTE To change a lens length to a precise value, double-click a camera glyph to open the Properties palette. In the Cameras section, Lens Length option (mm), enter a numeric value.

Command line: CAMERA

3D Navigate panel, Lens Length

To set a camera's clipping planes

To rename a camera

- 1 If cameras are not already displayed in the drawing, click View menu ▶ Display ▶ Cameras.

- 2** Double-click a camera glyph.
- 3** In the Properties palette, General section, Name option, enter a new name.
- 4** Press ENTER.

Command line: CAMERA

To change a camera's location

- 1** If cameras are not already displayed in the drawing, click View menu ➤ Display ➤ Cameras.
- 2** Click the camera glyph, drag it to the new location, and click to place the camera.
- 3** Press ENTER.

Command line: CAMERA

3D Navigate panel, Camera Position

To change a camera's target

- 1** If cameras are not already displayed in the drawing, click View menu ➤ Display ➤ Cameras.
- 2** Click the camera glyph whose target you want to change.
- 3** Click the Target Distance grip tool (the blue grip in the center), drag it to the new location, and click to place the target.
- 4** Press ENTER.

Command line: CAMERA

3D Navigate panel, Target Position

To specify whether the camera glyph plots

- 1** If cameras are not already displayed in the drawing, click View menu ➤ Display ➤ Cameras.
- 2** Double-click a camera.
- 3** In the Properties palette, Camera section, Plot option, click Yes or No.

Command line: CAMERA

Create Motion Path Animations

Motion path animations, such as 3D animated walk-throughs of a model, allow you to visually demonstrate a model to both a technical and non-technical audience. You can record and play back a navigation to communicate your design intent dynamically.

When you set up a motion path animation, you can:

Control a Camera Motion Path

You control the camera motion, and therefore the animation, by linking the camera and its target to a point or a path.

To create an animation using motion paths, you link the camera and its target to either a point or a path. If you want the camera to remain still, link it to a point. If you want the camera to move along a path, link it to a path.

If you want the target to remain still, link it to a point. If you want the target to move, link it to a path. You cannot link both the camera and the target to a point.

Use the same path when you want the animation view to follow in line with the camera's path. You do this by setting the target's path to None in the Motion Path Animation dialog box. This is the default setting.

NOTE To link a camera or target to a path, you must create the path object before you create the motion path animation. A path can be a line, arc, elliptical arc, circle, polyline, 3D polyline, or spline.

To create a motion path animation

- 1 In the drawing, create a path object for either the camera or the target. A path can be a line, arc, elliptical arc, circle, polyline, 3D polyline, or spline.

NOTE The path you create is not visible in the animation.

- 2 Click **View > Motion Path Animations**.
- 3 Do one of the following:
 - To specify a new camera point, click the **Pick Point** button, and specify a point in the drawing. Enter a name for the point. Click **OK**.
 - To specify a new camera path, click the **Select Path** button, and specify a path in the drawing. Enter a name for the path. Click **OK**.

- To specify an existing camera point or path, select it from the drop-down list.
- 4** In the Motion Path Animation dialog box, Target section, click either Point or Path.
- 5** Do one of the following:
- To specify a new target point, click the Pick Point button, and specify a point in the drawing. Enter a name for the point. Click OK.
 - To specify a new target path, click the Select Path button, and specify a path in the drawing. Enter a name for the path. Click OK.
 - To specify an existing target point or path, select it from the drop-down list.
- 6** In the Animation Settings section, adjust the animation settings to create the animation to suit your needs.
- 7** When you have finished adjusting the points, paths, and settings, click Preview to view the animation, or OK to save it.

Command line: ANIPATH

Specify Motion Path Settings

You determine the format of the animation file of a motion path animation by specifying settings in the Motion Path Animation dialog box.

Several settings control the animation's frame rate, duration, resolution, visual style, and file format.

To view a reverse motion path animation

- 1** Click View menu > Motion Path Animations.
- 2** In the Motion Path Animation dialog box, Animation Settings section, click the Reverse check box.
- 3** Click OK.

Command line: ANIPATH

To control the speed and duration of the animation

- 1** Click View menu > Motion Path Animations.

- 2** In the Motion Path Animation dialog box, Animation Settings section, specify a Frame rate (FPS).
- 3** Do one of the following:
 - Specify a number of frames.
 - Specify a duration, in seconds.
- 4** Click Preview or OK.

Command line: ANIPATH

To set animation resolution

- 1** Click View menu ► Motion Path Animations.
- 2** In the Motion Path Animation dialog box, Animation Settings section, select a resolution from the Resolution drop-down list.

Command line: ANIPATH

To set the video format

- 1** Click View menu ► Motion Path Animations.
- 2** In the Motion Path Animation dialog box, Animation Settings section, click a video format (AVI, MPG, MOV, or WMV) from the Format drop-down list.

Command line: ANIPATH

Record a Motion Path Animation

You can preview the animation before you record it and then save it in the desired format.

To preview and save a motion path animation

- 1** Click View menu ► Motion Path Animations.
- 2** In the Motion Path Animations dialog box, do the following:
 - Specify a point or path for the camera.
 - Specify a point or path for the target.
 - Adjust any animation settings.

- 3** To preview the animation, click the Preview button.
- 4** In the Animation Preview window, view the animation. When the preview animation is complete, close the Animation Preview window.
- 5** In the Motion Path Animations dialog box, click OK.
- 6** In the Save As dialog box, specify a file name and location to save the animation file.
- 7** Click Save.

Command line: ANIPATH

I 3

Display Multiple Views in Model Space

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

You can save arrangements of model space viewports for reuse at any time.

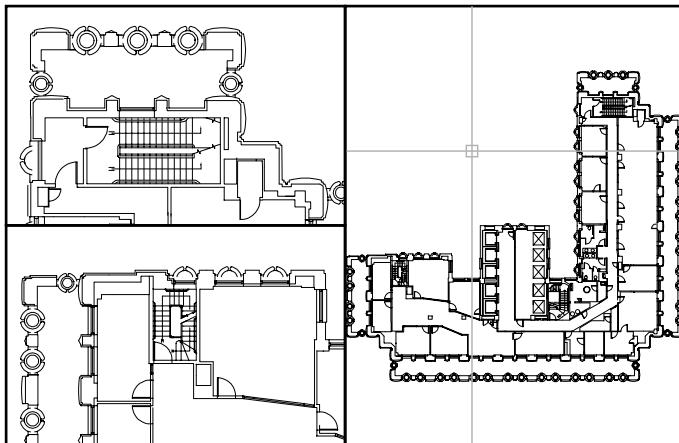
In this chapter

- Set Model Space Viewports
- Select and Use the Current Viewport
- Save and Restore Model Tab Viewport Arrangements

Set Model Space Viewports

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

Viewports created on the Model tab completely fill the drawing area and do not overlap. As you make changes in one viewport, the others are updated simultaneously. Three model space viewports are shown in the illustration.



You can also create viewports on a layout tab. You use those viewports, called *layout viewports*, to arrange the views of your drawing on a sheet. You can move and resize layout viewports. By using layout viewports, you have more control over the display; for example, you can freeze certain layers in one layout viewport without affecting the others. For more information about layouts and layout viewports, see “Create Multiple-View Drawing Layouts (Paper Space)” on page 209.

Use Model Space Viewports

With model space viewports, you can do the following:

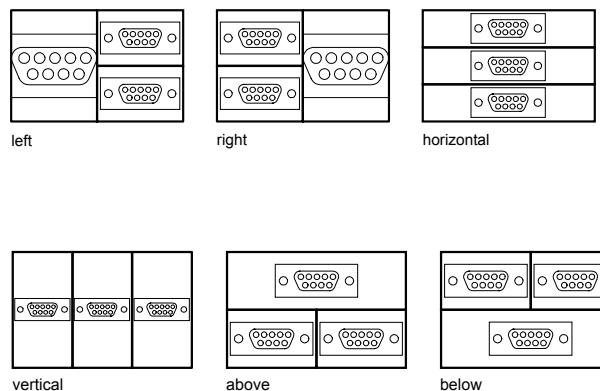
- Pan; zoom; set Snap, Grid, and UCS icon modes; and restore named views.
- Save user coordinate system orientations with individual viewports.

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

Setting up different coordinate systems in individual viewports is useful if you typically work on 3D models. See “Assign User Coordinate System Orientations to Viewports” on page 354.

Split and Join Model Space Viewports

The illustrations below show several default model space viewport configurations.



You can easily modify model space viewports by splitting and joining them. If you want to join two viewports, they must share a common edge of the same length.

To subdivide a viewport on the Model tab

- 1 If you have more than one viewport, click inside the viewport you want to subdivide.
- 2 Click View menu ► Viewports and then either 2 Viewports, 3 Viewports, or 4 Viewports to indicate how many model space viewports should be created.
- 3 At the Next prompt, specify the arrangement of the new viewports.

Command line: VPORTS

To join two viewports on the Model tab

- 1 Click View menu ► Viewports ► Join.
- 2 Click within the model space viewport containing the view you want to keep.
- 3 Click within an adjacent viewport to join it to the first viewport.

Command line: VPORTS

To restore a single viewport on the Model tab

- Click View menu ► Viewports ► 1 Viewport.

Command line: VPORTS

To switch from a layout tab to the Model tab

- Click the Model tab at the bottom of the drawing area.

Command line: TILEMODE

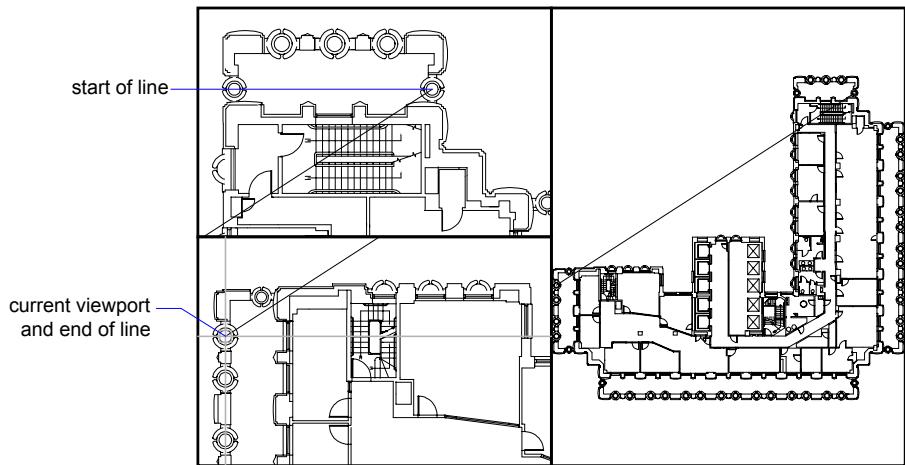
Select and Use the Current Viewport

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

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

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

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



To make a viewport current

- Click anywhere within the viewport border.

To cycle through viewports without clicking

- Press CTRL+ R repeatedly.

Save and Restore Model Tab Viewport Arrangements

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

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

Settings that are saved with viewport arrangements include

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

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

To save and name a viewport arrangement

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

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

Command line: VPORTS

To restore a saved viewport arrangement

- 1 Click View menu ► Viewports ► Named Viewports.
- 2 In the Viewports dialog box, Named Viewports tab, select the name of the viewport configuration from the list.
- 3 Click OK.

Command line: VPORTS

To delete a saved viewport arrangement

- 1 Click View menu ► Viewports ► Named Viewports.
- 2 In the Viewports dialog box, Named Viewports tab, select the name of the viewport configuration you want to delete.
- 3 Press DELETE.

Command line: VPORTS

To view a list of saved viewport arrangements

- Click View menu ► Viewports ► Named Viewports to display the Viewports dialog box.

All saved viewport arrangements in the drawing are listed on the Named Viewports tab under Named Viewports.

Command line: VPOTS

Part 4

Choose a Work Process Before You Begin

Chapter 14 Create Single-View Drawings (Model Space)

Chapter 15 Create Multiple-View Drawing Layouts (Paper Space)

Chapter 16 Work with Sheets in a Sheet Set

Create Single-View Drawings (Model Space)

4

In this chapter

- Quick Start for Model Space Drafting
- Draw, Scale, and Annotate in 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®.

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

Quick Start for Model Space Drafting

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

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

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

- It is suitable for 2D drawings only
- It does not support multiple views and view-dependent layer settings
- Scaling the annotation and title block requires computation

With this method, you always draw geometric objects at full scale (1:1) and text, dimensions, and other annotation at a scale that will appear at the correct size when the drawing is plotted.

See also:

- “Create Multiple-View Drawing Layouts (Paper Space)” on page 209
- “Work with Sheets in a Sheet Set” on page 239

Draw, Scale, and Annotate in Model Space

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

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

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

Determine the Unit of Measurement

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

Specify the Display Style of Drawing Units

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

Specify the display style of drawing units with the UNITS command. The default drawing unit type is decimal.

Set the Scale for Annotations and Blocks

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

You should enter the scale for the following objects:

- **Text.** Set the text height as you create text or by setting a fixed text height in the text style (STYLE).
- **Dimensions.** Set the dimension scale in a dimension style (DIMSTYLE) or with the DIMSCALE system variable.
- **Linetypes.** Set the scale for noncontinuous linetypes with the CELTSCALE and LTSCALE system variables.
- **Hatch patterns.** Set the scale for hatch patterns in the Hatch and Gradient dialog box (HATCH) or with the HPSCALE system variable.
- **Blocks.** Specify the insertion scale for blocks either as you insert them, or set an insertion scale in the Insert dialog box (INSERT) or in DesignCenter (ADCENTER). The system variables used for inserting blocks are INSUNITS, INSUNITSDEFSOURCE, and INSUNITSDEFTARGET. This also applies to the border and title block of the drawing.

Determine the Scale Factor for Plotting

To plot your drawing from the Model tab, you calculate the exact scale factor by converting the drawing scale to a ratio of 1: n . This ratio compares plotted

units to drawing units that represent the actual size of the objects you are drawing.

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

$$1/4" = 12"$$

$$1 = 12 \times 4$$

$$1 \text{ (plotted unit)} = 48 \text{ (drawing units)}$$

Using the same calculation, the scale factor for 1 centimeter = 1 meter is 100, and the scale factor for 1 inch = 20 feet is 240.

Sample Scale Ratios

The sample architectural scale ratios in the table can be used to calculate text sizes in model space.

Scale	Scale factor	To plot text size at	Set drawing text size to
1 cm = 1 m	100	3 mm	30 cm
1/8" = 1'-0"	96	1/8"	12"
3/16" = 1'-0"	64	1/8"	8"
1/4" = 1'-0"	48	1/8"	6"
3/8" = 1'-0"	32	1/8"	4"
1/2" = 1'-0"	24	1/8"	3"
3/4" = 1'-0"	16	1/8"	2"
1" = 1'-0"	12	1/8"	1.5"
1 1/2" = 1'-0"	8	1/8"	1.0"

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

$$210 \times 20 = 4200 \text{ mm}$$

$$297 \times 20 = 5900 \text{ mm}$$

See also:

["Specify Units and Unit Formats" on page 114](#)

To specify the display style for drawing units

- 1 Click Format menu ► Units.
- 2 In the Drawing Units dialog box, set the unit values for your drawing.
- 3 As you change unit settings, you can see examples under Sample Output.
 - Under Length, select a unit type and level of precision. This determines the display style for linear drawing units.
 - Under Drawing Units for DesignCenter Blocks, select the unit that you want used to scale blocks, images, or other content inserted into the drawing. If you do not want inserted content to be scaled, select Unitless.
 - Under Angle, select an angle type and precision. This determines the display style for angular drawing units. The default starting angle, 0 degrees, is toward 3 o'clock (or east).
 - To specify an angle direction, click Direction, and then select the base angle in the Direction Control dialog box. The angle direction controls the point from which angles are measured and the direction in which they are measured. If you select Other, you can enter an angle, or click Angle to specify an angle using your pointing device. The default positive angle measurement is counterclockwise.
- 4 Click OK to exit each dialog box.

Command line: UNITS

I 5

Create Multiple-View Drawing Layouts (Paper Space)

You *layout viewports* to display views of your model. Each view can have a different viewing angle, view scale, and layer display.

In this chapter

- Quick Start for Layouts
- Understand the Layout Process
- Work with Model Space and Paper Space
- Create and Modify Layout Viewports
- Control Views in Layout Viewports
- Reuse Layouts and Layout Settings

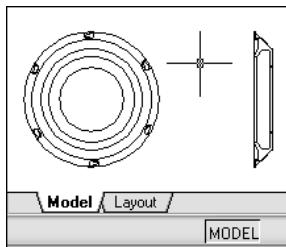
Quick Start for Layouts

There are two distinct working environments, or “spaces,” in which you can create objects in a drawing.

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

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

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



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

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

Understand the Layout Process

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

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

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

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

Process Summary

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

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

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

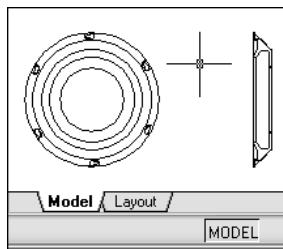
Work with Model Space and Paper Space

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

Work on the Model Tab

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

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



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

To activate the Model tab

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

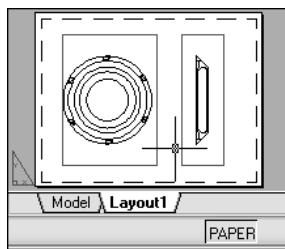
- Click the Model tab.
- Right-click any layout tab or the Model tab. Click Activate Model Tab.
- If the Model and layout tabs are hidden, click the Model button on the status bar at the bottom-center of the application window.

Work on a Layout Tab

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

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

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



NOTE These tabs can be hidden, appearing instead as buttons on the status bar at the bottom-center of the application window. To display the tabs, right-click the Model or layout button and click Display Layout and Model Tabs on the shortcut menu.

Create Additional Layout Tabs

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

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

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

Right-click a layout tab to display a layout shortcut menu with options.

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

Use the Layout Wizard to Specify Layout Settings

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

- A name for the new layout
- The printer associated with the layout
- A paper size to use for the layout
- The orientation of the drawing on the paper
- A title block
- Viewport setup information
- A location for the viewport configuration in the layout

You can edit the information entered in the wizard later by selecting the layout and clicking Page Setup on the File menu.

To specify the page setup for the current layout

- Right-click the current layout tab. Click Page Setup.

Layouts toolbar



Command line: PAGESETUP

To create a new layout

- 1 Click Insert menu ▶ Layout ▶ New Layout.
- 2 Enter the name of the new layout on the command line.
A new layout tab is created. To switch to the new layout, choose the layout tab.

Layouts toolbar



Command line: LAYOUT

To import a layout from a template

- 1 Click Insert menu ▶ Layout ▶ Layout from Template.
- 2 In the Select File dialog box, select a DWT or DWG file to import a layout from.
- 3 Click Open.
- 4 In the Insert Layout(s) dialog box, select a layout to import.
A new layout tab is created. To switch to the new layout, click the layout tab.

Layouts toolbar



Command line: LAYOUT

To create a layout using a wizard

- 1 Click Insert menu ► Layout ► Layout Wizard.
- 2 On each page of the Create Layout wizard, select the appropriate settings for the new layout.

When finished, the new layout will be the current layout tab.

Command line: LAYOUTWIZARD

To duplicate a layout

- 1 Right-click the layout tab of the layout you want to duplicate. Click Move or Copy.
- 2 In the Move or Copy dialog box, select a position for the new layout tab.
- 3 Make sure that Create a Copy is selected.
- 4 Click OK.

NOTE The Model tab cannot be duplicated.

Command line: LAYOUT

To rename a layout

- 1 Right-click the layout tab for the layout you want to rename. Click Rename.
- 2 In the Rename Layout dialog box, enter the new name for the layout.
- 3 Click OK.

NOTE The Model tab cannot be renamed.

Command line: LAYOUT

To delete a layout

- 1 Right-click the layout tab for the layout you want to delete. Click Delete.
- 2 In the warning box, click OK to delete the layout.

Any named views associated with the layout are deleted automatically when the layout is deleted.

NOTE The Model tab cannot be deleted.

Command line: LAYOUT

To rearrange layout tabs

- 1 Right-click the layout tab of the layout you want to reposition. Click Move or Copy.
- 2 In the Move or Copy dialog box, select the layout tab that should be placed immediately after the moved or copied layout tab. If you want to move the layout tab to the end of the list of layout tabs, select Move to End.
- 3 Click OK.

NOTE The Model tab cannot be repositioned.

Command line: LAYOUT

To make a layout current

Do one of the following to make a layout current:

- Click the layout tab for the layout that you want to make current.
- Press CTRL+Page Down to cycle through the layout tabs from left to right or CTRL+Page Up to cycle through the layout tabs from right to left. Stop when you get to the tab of the layout that you want to make current.

To move an object from model space to paper space (or vice versa)

- 1 Click Modify menu > Change Space.
- 2 Select one or more objects to move.
- 3 Press ENTER to end the command.

Command line: CHSPACE

To activate the previous layout

- 1 Right-click any layout tab or the Model tab.
- 2 Click Activate Previous Layout.

To select all layouts

- Right-click any layout tab. Click Select All Layouts.

Command line: LAYOUT

To plot a layout

- 1 Do *one* of the following:
 - Right-click a layout tab. Click Plot.
 - Hold down SHIFT to select several layout tabs. Right-click a layout tab. Click Publish Selected Layouts.
- 2 Select settings in the Plot or the Publish dialog box. Click OK or Publish.

Standard toolbar



Command line: PLOT

Access Model Space from a Layout Viewport

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

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

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

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

Create and Modify Objects in a Layout Viewport

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

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

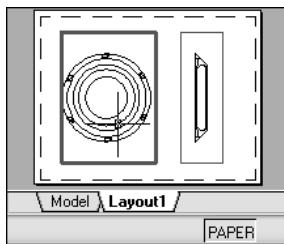
NOTE If you use PLOT while a viewport is maximized, the layout tab is restored before the Plot dialog box is displayed. If you save and close the drawing while a viewport is maximized, the drawing opens with the layout tab restored.

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

Adjust the View in a Layout Viewport

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

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



To switch between model space and paper space on a layout

On a layout, use one of the following methods:

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

You are now in paper space. You can create and modify objects on the layout.

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

To edit in a maximized layout viewport

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

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

- 2 On the status bar, click the Maximize Viewport button.

You can restore the viewport and maximize another viewport by clicking one of the arrows next to the Maximize Viewport button.

- 3 Make any changes.

- 4 To return to the layout viewport, click the Restore Viewport button on the status bar.

The center point and magnification are returned to the settings that were in effect before the viewport was maximized.

Status bar



Command line: VPMAX, VPMIN

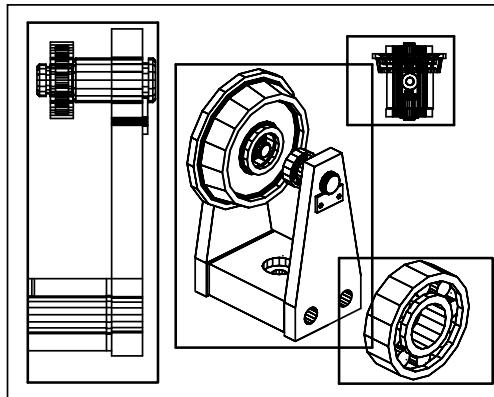
Shortcut menu: Right-click. Click Maximize Viewport.

Create and Modify Layout Viewports

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

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

With 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.

The MVIEW command provides two options to assist you in defining an irregularly shaped viewport.

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

The Polygonal option can be used to create a nonrectangular layout viewport by specifying points. The prompt sequence is the same as the prompt sequence for creating a polyline.

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

Resize Layout Viewports

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

Clip Layout Viewports

You can redefine the boundary of a layout viewport by using the VPCLIP command. To clip a layout viewport, you can use your pointing device to either select an existing object to designate as the new boundary, or specify the points of a new boundary.

To create a new layout viewport

- 1 On the layout tab, click View menu ► Viewports ► 1 Viewport.
- 2 Click to specify one corner of the new layout viewport.
- 3 Click to specify the opposite corner.

A new layout viewport object is available and displays a default view. To adjust the view, double-click within the layout viewport to access model space.

Viewports toolbar



Command line: VPORTS, MVIEW

To create a viewport configuration on a layout

- 1 Click a layout tab.
- 2 Click View menu ► Viewports ► New Viewports.
- 3 In the Viewports dialog box, New Viewports tab, select a viewport configuration from the list.
- 4 Under Setup, select either 2D or 3D.
When you select 3D, a set of standard 3D views is applied to each viewport in the configuration.
- 5 Under Viewport Spacing, select the amount of spacing you want to add between the viewports.
- 6 To change a view, select a viewport in the preview image. Under Change View To, select a view from the list of standard views.
The list includes top, bottom, front, back, left, right, and isometric views, along with any named views that are saved in the drawing. The selected view is displayed under Preview.
- 7 Click OK.

- 8** In the drawing area, specify two points to indicate the area to contain the viewport configuration.

Viewports toolbar



Command line: VPORTS

To place a named viewport configuration into a layout

- 1** Click a layout tab.
- 2** Click View menu > Viewports > New Viewports.
- 3** In the Viewports dialog box, Named Viewports tab, select the named viewport configuration from the list.
- 4** Click OK.
- 5** In the layout, specify a location for the named viewport configuration.

Viewports toolbar



Command line: VPORTS

To modify viewport properties using the Properties palette

- 1** Click the border of the layout viewport whose properties you want to modify.
- 2** Right-click, and then click Properties.
- 3** In the Properties palette, select the value for the property you want to modify. Enter a new value or select a new setting from the list provided.
The new property setting or value is assigned to the current viewport.

Standard toolbar



Command line: PROPERTIES

Shortcut menu: Select the viewport and right-click in the drawing area. Click Properties.

To clip a viewport boundary

- 1 At the Command prompt, enter **vpclip**.
- 2 Select the viewport to clip.
- 3 Enter **d** (Delete) to delete the clipping boundary.
- 4 Enter **p** (Polyangular).
- 5 Specify points, or select the object, to define the new viewport boundary.

Command line: Select the viewport to clip and right-click in the drawing area. Click Viewport Clip.

Control Views in Layout Viewports

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

Scale Views in Layout Viewports

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

You can change the view scale of the viewport using

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

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

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

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

Lock the Scale of Layout Viewports

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

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

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

To modify a layout viewport scale using the Properties palette

- 1 Make sure you are on a layout tab in paper space.
- 2 Click the border of the viewport whose scale you want to modify.
- 3 Right-click, and then click Properties.
- 4 In the Properties palette, select Standard Scale, and then select a new scale from the list.

The scale you choose is applied to the viewport.

NOTE To use a custom scale, enter a scale in the Custom Scale field in the Properties palette.

Standard toolbar



Command line: PROPERTIES

Shortcut menu: Select the viewport and right-click in the drawing area. Click Properties.

To turn on scale locking in a layout viewport

- 1 In the layout, click the viewport whose scale you want to lock.

- 2** If necessary, open the Properties palette.
- 3** In the Properties palette, do one of the following:
 - If you selected a rectangular viewport, select Display Locked, and then click Yes.
 - If you selected a nonrectangular viewport, first click All (2) and select Viewport (1). Then select Display Locked, and click Yes.

The current viewport's scale is locked. If you change the zoom factor in the viewport, only paper space objects are affected.

Standard toolbar



Command line: PROPERTIES

Shortcut menu: Select the viewport and right-click in the drawing area. Click Properties.

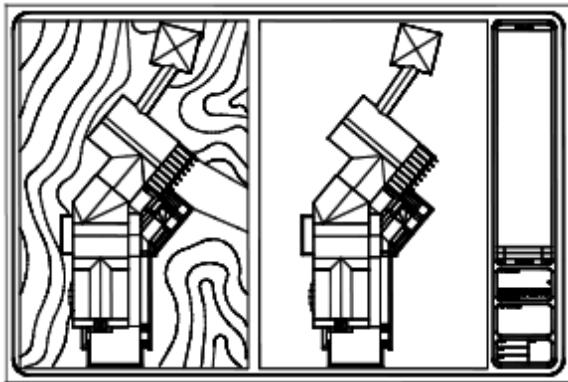
Control Visibility in Layout Viewports

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

Freeze Specified Layers in a Layout Viewport

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

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



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

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

Freeze or Thaw Layers Automatically in New Layout Viewports

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

Create New Layers That Are Frozen in All Layout Viewports

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

To freeze or thaw layers in the current layout viewport

- 1 Double-click within a layout viewport to make it current.
- 2 Click Format menu ► Layer.
- 3 In the Layer Properties Manager, select the layers to freeze or thaw.
Hold down CTRL to select more than one layer. Hold down SHIFT to select a sequence of layers.

- 4 Click the icon in the Current VP Freeze column for one of the selected layers.
- 5 Click OK.

Layers toolbar



Command line: LAYER

To view a list of layers that are frozen in the current viewport

- 1 Click a layout tab.
- 2 Double-click within a layout viewport to make it current.
- 3 Click Format menu ► Layer.
- 4 In the Layer Properties Manager, look at the Current VP Freeze column for the Freeze/Thaw in Current Viewports icon.



- 5 Click OK.

Layers toolbar



Command line: LAYER, VPLAYER

To freeze or thaw layers in all viewports

- 1 Click a layout tab.
- 2 Click Format menu ► Layer.
- 3 In the Layer Properties Manager, select one or more layers to freeze or thaw.
Hold down CTRL to select more than one layer. Hold down SHIFT to select a sequence of layers.
- 4 In the Freeze column, click the icon to freeze or thaw.

Layers toolbar



Command line: LAYER

To freeze or thaw layers in paper space

- 1 Click a layout tab.
- 2 Make sure that you are in paper space. (On the status bar, PAPER is on.)
- 3 Click Format menu ► Layer.
- 4 In the Layer Properties Manager, select the layer or layers you want to freeze or thaw.
- 5 In the Freeze column, click the icon to change a layer's state. The sun icon means that a layer is thawed; the snowflake icon means that a layer is frozen.



- 6 Click OK.

Layers toolbar



Command line: LAYER

To freeze or thaw layers in all new viewports

- 1 Click a layout tab.
- 2 Click Format menu ► click Layer.
- 3 In the Layer Properties Manager, select the layer or layers you want to be automatically frozen or thawed in new viewports you create.
Hold down CTRL to select more than one layer. Hold down SHIFT to select a sequence of layers.
- 4 In the New VP Freeze column, click the icon to change a layer's state. The sun icon means that a layer is thawed; the snowflake icon means that a layer is frozen.
- 5 Click OK.

Layers toolbar



Command line: LAYER

To create new layers that are frozen in all viewports

- 1 Click a layout tab.
- 2 Click Format menu ► Layer.
- 3 Click the New Layer button to create a layer.
- 4 Rename the new layer.
- 5 Click the icon in the Freeze column to change the layer's state to frozen. The sun icon means that a layer is thawed; the snowflake icon means that a layer is frozen.
- 6 Click OK.

Layers toolbar



Command line: LAYER, VPLAYER

Screen Objects in Layout Viewports

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

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

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

See also:

[“Set Options for Plotted Objects” on page 990](#)

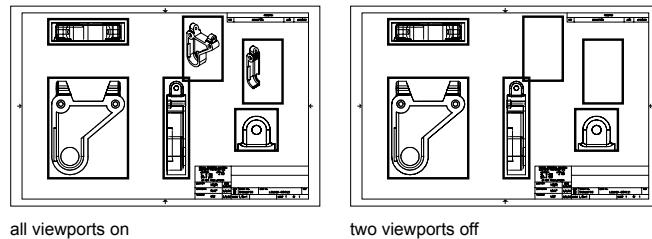
To apply screening to objects in a layout viewport

- 1 Click File menu ► Plot Style Manager.
- 2 Right-click a CTB or STB file. Click Open.
- 3 In the Plot Style Table Editor, Form View tab, select a plot style to change.
- 4 In the Screening box, enter an intensity value between 1 and 100.
- 5 Click Save & Close.
- 6 In the layout, click File menu ► Page Setup.
- 7 In the Page Setup dialog box, Plot Device tab, select the edited plot style table from the Plot Style Table (Pen Assignments) list.
- 8 Double-click within the layout viewport that contains the objects whose screening you want to change.
- 9 Select the objects whose plot style you want to change.
- 10 Right-click in the drawing area and click Properties.
- 11 In the Properties palette, use one of the following methods:
 - If you are using named plot style tables, next to Plot Style, select the plot style you edited in the Plot Style Table Editor. If that plot style is not listed, select Other and set Active Plot Style Table to the plot style you edited in the Plot Style Table Editor. Select the edited plot style from the Plot Styles list in the Select Plot Style dialog box.
 - If you are using color-dependent plot style tables, next to Color, select the color whose plot style you edited in the Plot Style Table Editor.

Turn Layout Viewports On or Off

You can save time by turning some layout viewports off or by limiting the number of active viewports.

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



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

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

To turn viewports on or off using the Properties palette

- 1 Make sure you are on a layout tab in paper space.
- 2 Click the border of the viewport to turn on or off.
- 3 Right-click, and then click Properties.
- 4 In the Properties palette, under Misc, select On, and then select Yes or No to turn the viewport on or off.

For a nonrectangular viewport, select All (2) in the Properties palette, and then select Viewport (1) before changing any viewport properties.

Standard toolbar



Command line: PROPERTIES

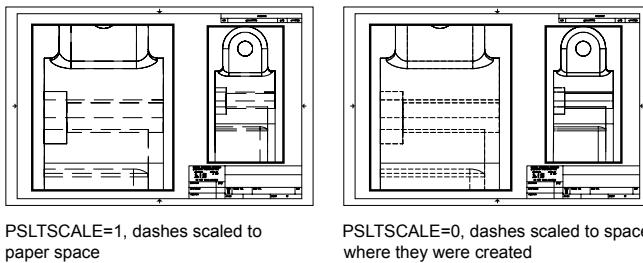
Shortcut menu: Select the viewport and right-click in the drawing area. Click Display Viewport Objects.

Scale Linetypes in Layout Viewports

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

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

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



PSLTSCALE=1, dashes scaled to paper space

PSLTSCALE=0, dashes scaled to space where they were created

See also:

[“Set the Lineweight Scale for a Layout” on page 958](#)

To scale linetypes globally in paper space

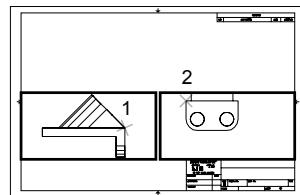
- 1 Click Format menu ► Linetype.
- 2 In the Linetype Manager, click Show Details.
- 3 Under Global Scale Factor, enter a global scale to apply to the linetypes.
- 4 Click OK.

Command line: LINETYPE

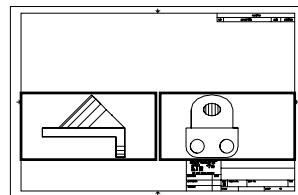
Align Views in Layout Viewports

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

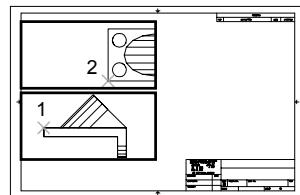
For angled, horizontal, and vertical alignments, you can move each layout viewport relative to distances defined by the model-space geometry displayed.



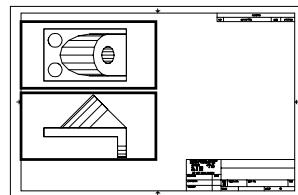
Horizontal alignments



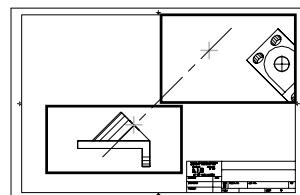
Horizontal alignments



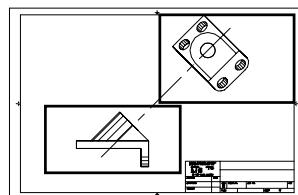
Vertical alignments



Vertical alignments



Angled alignments



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

To align objects between viewports using a construction line

- 1 Make sure you are on a layout tab.
- 2 Click Draw menu ▶ Construction Line.
- 3 Specify a point in the first viewport. Specify a second point to determine a line for the alignment.
Choose a point that can be aligned with objects in the second viewport. Use object snaps for precision.
- 4 Click Modify menu ▶ Move.
- 5 Select the viewport to align to the first viewport. Press ENTER.
- 6 When prompted for a base point, specify a point in the second viewport. Choose a point that corresponds with the point selected in the first viewport.
- 7 When prompted for the second point, hold down SHIFT and right-click. Choose Object Snap menu ▶ Perpendicular. Click on the construction line you created.

The first and second viewports, and the objects in the viewports, are aligned.

NOTE When aligning objects in viewports, the scale of the viewports should be the same.

Modify toolbar



To align objects between viewports using MVSETUP

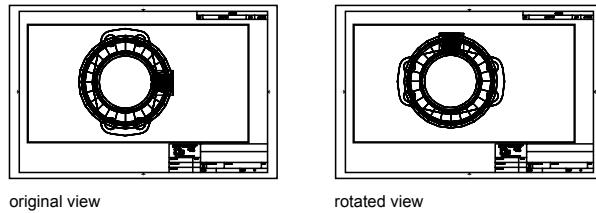
- 1 At the Command prompt, enter **mvsetup**.
- 2 Enter **a** (Align).
- 3 Select one of the following alignments:
 - **Horizontal**. Aligns a point in one viewport horizontally with a base point in another viewport.
 - **Vertical**. Aligns a point in one viewport vertically with a base point in another viewport.
 - **Angled**. Aligns a point in one viewport at a specified distance and angle from a base point in another viewport.

- 4 Make sure the viewport with the view that is to remain stationary is current. Then specify a base point.
- 5 Select the viewport with the view you want to realign. Then specify an alignment point in that view.
- 6 For angled alignments only, specify a distance and displacement angle from the base point to the alignment point in the second viewport.

Rotate Views in Layout Viewports

You can rotate an entire view within a layout viewport by changing the UCS and using the PLAN command.

With the 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.



For a faster method, use the Align and then Rotate View options in the MVSETUP command.

NOTE The ROTATE command rotates individual objects only and should not be used to try to rotate a view.

To rotate a view by changing the UCS

- 1 Make sure you are on a layout tab.
- 2 Double-click within the viewport whose objects you want to rotate.
- 3 Make sure that the current UCS is parallel to the plane of rotation (the UCS icon should look normal). If the UCS is not parallel to the plane of rotation, click Tools menu ▶ New UCS ▶ View.
- 4 Click Tools menu ▶ New UCS ▶ Z. To rotate the view 90 degrees clockwise, enter **90**. To rotate the view 90 degrees counter-clockwise, enter **-90**.

5 Click View menu ► 3D Views ► Plan View. Click Current UCS.

The entire view rotates within the viewport. You may need to respecify the scale of the viewport.

UCS toolbar



Command line: UCS

To rotate a view using MVSETUP

- 1 At the Command prompt, enter **mvsetup**.
- 2 Enter **a** (Align).
- 3 Enter **r** to rotate the view.
- 4 Select the viewport with the view you want to rotate.
- 5 Specify a base point for the rotation.
- 6 Specify the rotation angle.

The entire view rotates within the viewport.

Reuse Layouts and Layout Settings

When you create a layout, you can choose to apply the information from an existing template.

A layout template is a layout imported from a DWG or DWT file. When you create a layout, you can choose to apply the information from an existing template. The program has sample layout templates to use when you design a new layout environment. The paper space objects and page setup in the existing template are used in the new layout. Thus, the layout objects, including any viewport objects, are displayed in paper space. You can keep any of the existing objects from the template you import, or you can delete the objects. No model space objects are imported.

The layout templates are identified with a *.dwt* file extension. However, a layout template or layout from any drawing or drawing template can be imported into the current drawing.

Save a Layout Template

Any drawing can be saved as a drawing template (DWT file), including all of the objects and layout settings. You can save a layout to a new DWT file by

choosing the Save As option of the LAYOUT command. The template file is saved in the drawing template file folder as defined in the Options dialog box, Support tab. The layout template has a *.dwt* or *.dwg* extension like a drawing template or drawing file, but it contains little information not essential to the layout.

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

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

Insert a Layout Using DesignCenter

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

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

To create a layout using a layout template

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

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

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

Layouts toolbar



Command line: LAYOUT

Shortcut menu: Right-click the layout tab. Click From Template.

To save a layout template

- 1 At the Command prompt, enter **layout**.
- 2 At the prompt, enter **sa** to save the current layout as a template.
- 3 Enter the name of the layout you are saving.
- 4 In the Create Drawing File dialog box, enter a name for the drawing template file you are saving.
- 5 In Files of Type, select Drawing Template File (*.dwt).
- 6 Click Save.

To insert a layout using DesignCenter

- 1 Click Tools menu ► DesignCenter.
- 2 In the tree view, find the drawing that contains the layout you want to reuse.
- 3 Double-click the drawing name to expand the options beneath it.
- 4 Select the Layouts icon to display the individual layouts in the content area.
- 5 Use one of the following methods to insert the layout into the current drawing:
 - Drag the layout icon from the content area into the drawing.
 - Select a layout in the content area and right-click. Click Add Layout(s).
 - Double-click the layout in the content area.

Standard toolbar



Command line: ADCENTER

Work with Sheets in a Sheet Set

| 6

With the Sheet Set Manager, you can organize drawing layouts into named sheet sets. The sheets in a sheet set can be transmitted, published, and archived as a unit.

Sheet sets facilitate the organization and management of drawings in a project, and improve the communication in a work group.

In this chapter

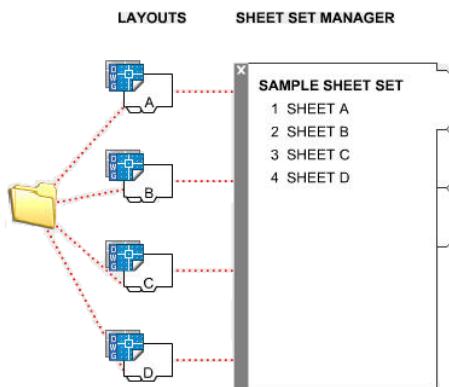
- Quick Start for Sheet Sets
- Understand the Sheet Set Manager Interface
- Create and Manage a Sheet Set
- Publish, Transmit, and Archive Sheet Sets
- Use Sheet Sets in a Team

Quick Start for Sheet Sets

A sheet set is an organized collection of *sheets* from several drawing files. A sheet is a selected layout from a drawing file.

Sets of drawings are the primary deliverable for most design groups. Sets of drawings communicate the overall design intent of a project and provide the documentation and specifications for the project. However, managing sets of drawings manually can be complicated and time consuming.

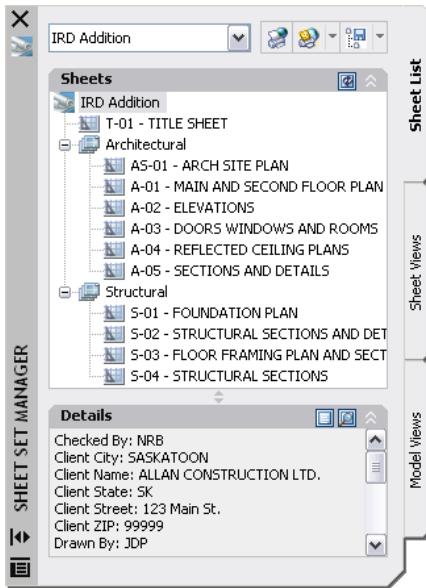
With the Sheet Set Manager, you can manage drawings as *sheet sets*. A sheet set is an organized and named collection of sheets from several drawing files. A sheet is a selected layout from a drawing file. You can import a layout from any drawing into a sheet set as a numbered sheet.



You can manage, transmit, publish, and archive sheet sets as a unit.

Understand the Sheet Set Manager Interface

Using the controls in the Sheet Set Manager, you can create, organize, and manage sheets in a sheet set.



You can use the following tabs and controls in the Sheet Set Manager:

Sheet Set Control. Lists menu options to create a new sheet set, open an existing sheet set, or switch between open sheet sets.

Sheet List Tab. Displays an organized list of all sheets in the sheet set. Each sheet in a sheet set is a specified layout in a drawing file.

Sheet Views Tab. Displays an organized list of all sheet views in the sheet set. Only sheet views created with AutoCAD 2005 and later are listed.

Model Views Tab. Lists the paths and folder names for the drawings containing model space views to be used in the sheet set.

- Click a folder to list the drawing files that are located in that folder.
- Click a drawing file to list the named model space views that are available for placement in the current sheet.
- Double-click a view to open the drawing containing the view.
- Right-click or drag a view to place it in the current sheet.

Buttons. Provides convenient access for the most commonly used operations for the currently selected tab.

Tree View. Displays the contents of a tab.

Details or Preview. Displays either descriptive information or a thumbnail preview of the currently selected item in the tree view.

Actions Used in the Tree View

You can use the following actions in the tree view:

- Right-click to access shortcut menus of the operations that are relevant to the currently selected item.
- Double-click items to open them. This is a convenient method for opening drawing files from the Sheet List tab or from the Model Views tab. You also double-click items in the tree view to expand or collapse them.
- Click one or more items to select them for operations such as opening, publishing, or transmitting.
- Click a single item to display descriptive information or a thumbnail preview of a selected sheet, view, or drawing file.
- Drag items within the tree view to reorder them.

NOTE To use the Sheet Set Manager effectively, right-click items in the tree view to access relevant shortcut menus. For access to shortcut menus in the drawing area that are needed for sheet set operations, the Shortcut Menus in Drawing Area must be checked in the Options dialog box, User Preferences tab.

Create and Manage a Sheet Set

There are several methods for setting up and organizing a sheet set. You can also include relevant information with a sheet set and its components.

Create a Sheet Set

You create a sheet set with the Create a Sheet Set wizard. In the wizard, you can either create a sheet set from scratch based on existing drawings or use an example sheet set as a template.

Layouts from specified drawing files are imported into the sheet set. The associations and information that define a sheet set are stored in a sheet set data (DST) file.

When creating a new sheet set using the Create Sheet Set wizard, a new folder is created as the default sheet set storage location. This new folder, named *AutoCAD Sheet Sets*, is located in the *My Documents* folder. The default location

for the sheet set file can be changed, and it is recommended that the DST file is stored with the project files.

NOTE The DST file should be stored in a network location that is accessible to all sheet set users on the network and mapped using the same logical drive. It is strongly recommended that you store the DST and the sheet drawings in the same folder. If an entire sheet set needs to be moved, or a server or folder name changes, the DST file will still be able to locate the sheets using relative path information.

Preparation Tasks

Before you begin creating a sheet set, you should complete the following tasks:

- **Consolidate drawing files.** Move the drawing files to be used in the sheet set into a small number of folders. This will simplify sheet set administration.
- **Eliminate multiple layout tabs.** Each drawing you plan to use in the sheet set should have only one layout to be used as a sheet in the sheet set. This is important for access to sheets by multiple users. Only one sheet in each drawing can be open at a time.
- **Create a sheet creation template.** Create or identify a drawing template (DWT) file to be used by the sheet set for creating new sheets. This drawing template file is called the *sheet creation template*. You specify this template file in the Sheet Set Properties dialog box or the Subset Properties dialog box.
- **Create a page setup overrides file.** Create or identify a DWT file to store page setups for plotting and publishing. This file, called the *page setup overrides file*, can be used to apply a single page setup to all sheets in a sheet set, overriding the individual page setups stored in each drawing.

NOTE Although it is possible to use several layouts from the same drawing file as separate sheets in a sheet set, it is not recommended. This makes concurrent access to each layout by multiple users impossible. This practice can also reduce your management options and can complicate the organization of your sheet sets.

Create a Sheet Set from an Example Sheet Set

In the Create Sheet Set wizard, when you choose to create a sheet set from an example, the example sheet set provides the organizational structure and default settings for the new sheet set. You can also specify that folders are created corresponding to the subset storage paths of the sheet set.

After you create an empty sheet set with this option, you can import layouts or create sheets individually.

Create a Sheet Set from Existing Drawing Files

In the Create Sheet Set wizard, when you choose to create a sheet set from existing drawing files, you specify one or more folders that contain drawing files. With this option, you can specify that the subset organization for the sheet set duplicates the folder structure of the drawing files. The layouts from these drawings can be imported into the sheet set automatically.

You can easily add more folders containing drawings by clicking the Browse button for each additional folder.

Backup and Recover Sheet Set Data Files

The data stored in the sheet set data file represents a significant amount of work, so you should take the same care to create backups of DST files as you do for drawing files.

In the unlikely event of DST file corruption or a major user error, the previously saved sheet set data file can be recovered. Every time the sheet set data file is opened, the current sheet set data file is copied to a backup file (DS\$). This backup file has the same file name and is located in the same folder as the current sheet set data file.

To recover the previous version of the sheet set data file, first make sure that no one else on your network is working on the sheet set. Then, it is recommended that you copy the existing DST file to another file name. Finally, rename the backup file from the DS\$ file extension to the DST file extension.

To open the Sheet Set Manager

- Click Tools menu ► Palettes ► Sheet Set Manager.

Standard toolbar



Command line: SHEETSET

To create a new sheet set

- 1 To create a new sheet set, do *one* of the following:
 - Click File menu ► New Sheet Set.
 - In the Sheet Set Manager, click the Sheet Set control. Click New Sheet Set.
- 2 Follow the steps in the Create Sheet Set wizard.

Standard toolbar



Command line: NEWSHEETSET

To open a sheet set

- 1 To open a sheet set, do *one* of the following:
 - Click File menu ▶ Open Sheet Set.
 - In the Sheet Set Manager, click the Sheet Set control. Click Open.
 - Double-click a sheet set data (DST) file.
- 2 In the Open Sheet Set dialog box, navigate to a folder that contains a DST file. Click the DST file and then click open.

The Sheet Set Manager displays the sheet set data.

NOTE You can open several sheet sets and then use the Sheet Set control to switch between them.

Standard toolbar



Command line: SHEETSET

To close a sheet set

- 1 In the Sheet Set Manager, Sheet List tab, right-click the sheet set node (at the top of the list). Click Close Sheet Set.

The sheet set is no longer displayed in the Sheet Set Manager.

NOTE Close a sheet set to reduce the number of sheet sets listed in the Sheet Set control. However, if you close a sheet set while you still have sheets in the sheet set open, you will not be able to update the Sheet List table and certain fields.

Standard toolbar

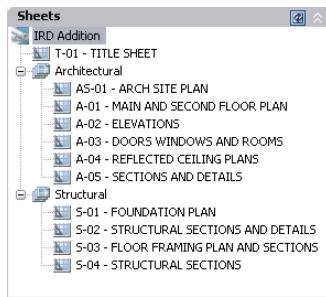


Command line: SHEETSET

Organize a Sheet Set

You can organize a sheet set by creating a hierarchy of subsets and categories. Sheets can be arranged into collections called subsets. Views can be arranged into collections called categories.

With a large sheet set, you will find it essential to organize sheets and views in the tree view. On the Sheet List tab, sheets can be arranged into collections called *subsets*. On the Sheet Views tab, views can be arranged into collections called *categories*.



Use Sheet Subsets

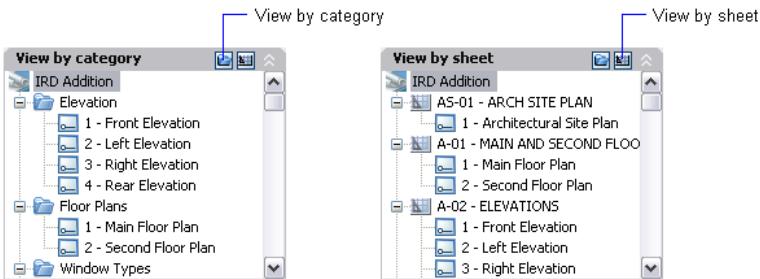
Sheet subsets are often associated with a discipline such as architecture or mechanical design. For example, in architecture, you might use a subset named Structural; and in mechanical design, you might use a subset called Standard Fasteners. In some cases, you might also find it useful to create subsets associated with a review or completion status.

You can nest subsets into other subsets as needed. After you create or import sheets or subsets, you can reorder them by dragging them in the tree view.

Use View Categories

View categories are often associated with a function. For example, in architecture, you might use a view category called Elevations; and in mechanical design, you might use a view category called Exploded.

You can display views either by category or by the sheet on which they are located.



You can nest categories into other categories as needed. To move a view to a different category, drag them in the tree view or use the Set Category shortcut menu option.

To reorder sheets in the sheet list

- In the Sheet Set Manager, on the Sheet List tab, drag a sheet up or down the list.

The sheet is relocated above or below its previous location on the sheet list. The sheet may also be moved into or out of a sheet subset.

Standard toolbar



Command line: SHEETSET

To create a new subset in the sheet list

- 1 In the Sheet Set Manager, Sheet List tab, right-click the sheet set node (at the top of the list), or an existing subset. Click New Subset.
- 2 In the Subset Properties dialog box, under Subset Name, enter the name of the new subset. Click OK.

You can drag the new subset anywhere on the sheet list, even under other subsets.

NOTE If you want to create a subset under an existing subset, you can right-click the existing subset. On the shortcut menu, click New Subset.

Standard toolbar



Command line: SHEETSET

To create a new view category in the view list

- 1 In the Sheet Set Manager, Sheet Views tab, click the View by Category button.
- 2 Right-click the sheet set node (at the top of the list). Click New View Category.
- 3 In the View Category dialog box, under Category Name, enter the name of the new view category.
- 4 If a list of blocks is displayed, select the callout blocks to be used for the views in this view category. You can add blocks to the list by clicking the Add Blocks button.

Standard toolbar



Command line: SHEETSET

To remove a subset from the sheet list

- 1 In the Sheet Set Manager, Sheet List tab, drag all sheets out of the subset that you want to remove.
- 2 Right-click the subset that you want to remove. Click Remove Subset.

Standard toolbar



Command line: SHEETSET

To remove a view category from the view list

- 1 In the Sheet Set Manager, Sheet Views tab, click the View by Category button.
- 2 Drag all views out of the view category that you want to remove.

- 3 Right-click the view category that you want to remove. Click Remove Category.

Standard toolbar



Command line: SHEETSET

Create and Modify Sheets

There are several options in the Sheet Set Manager for creating sheets and adding views, either through a shortcut menu or one of the tab buttons.

Following are descriptions of common sheet operations. You can access commands by right-clicking an item in the tree view to display the relevant shortcut menu.

- **Import layout as sheet.** After you create a sheet set, you can import one or more layouts from existing drawings. This is a fast method for creating multiple sheets from layouts in several drawings.
- **Create a new sheet.** As an alternative to importing existing layouts, you can create a new sheet. When you place views in this sheet, the drawing files associated with the views are attached as xrefs to the sheet drawing. The sheet drawing file is created using either the AutoCAD 2004 format or the AutoCAD 2007 format, depending on the format specified on the Open and Save tab in the Options dialog box.
- **Modify a sheet.** Double-click a sheet on the Sheet List tab to open a drawing from the sheet set. Use SHIFT or CTRL to select multiple sheets. To review a sheet, use the shortcut menu to open a drawing in read-only mode.

NOTE Modifying a sheet should always be done using an open sheet set in the Sheet Set Manager. This ensures that all data associated with the sheet are updated.

- **Rename and renumber a sheet.** After you create a sheet, you can change the sheet title and the sheet number. You can also specify a different drawing file associated with the sheet.
- **Remove a sheet from a sheet set.** Removing a sheet from a sheet set disassociates the sheet from the sheet set, but does not delete the drawing file or the layout.

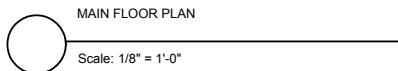
- **Reassociate a sheet.** If you move a sheet to a different folder, you should reassociate the sheet to the sheet set with the Sheet Properties dialog box to correct the path. For any relocated sheet drawing, the paths for Expected Layout and Found Layout are displayed in the Sheet Properties dialog box. To reassociate the sheet, click the path in Expected Layout and then click to navigate to the new location of the sheet.

NOTE You can quickly confirm whether a sheet is in the expected folder by looking at Details at the bottom of the Sheet List tab. If the selected sheet is not in the expected location, path information for both Expected Location and Found Location is displayed in Details.

- **Add a view to a sheet.** From the Model Views tab, you can easily add a view to a sheet by placing a named model space view or the entire drawing onto the current sheet.

NOTE After creating a named model space view, you must save the drawing to add the view to the Model Views tab. Click Refresh on the Model Views tab to update the Sheet Set Manager tree view.

- **Add label blocks to views.** With the Sheet Set Manager, you can label views and details automatically as you place them. Labels contain data associated with the referenced view.



- **Add callout blocks to views.** *Callout blocks* is the term for the symbols that reference other sheets. Callout blocks have many industry-specific names such as reference tags, detail keys, detail makers, building section keys, and so on. Callout blocks contain data associated with the sheet and view that are referenced.

NOTE When you place a callout block with fields or a view on a sheet, make sure that the current layer is unlocked.



■ **Create a title sheet and table of contents.** The first sheet in a sheet set will usually be a title sheet that includes a description of the sheet set and a table that lists all the sheets in the sheet set. You can create this table, called a *sheet list table*, on an open sheet. The table automatically includes all the sheets in the sheet set. You can create a sheet list table from the sheet set level shortcut menu only when a sheet is open. Once a sheet list table is created, you also have options to edit, update, or delete the cell content of the table.

NOTE For access to shortcut menus in the drawing area that are needed for sheet list table operations, the Shortcut Menus in Drawing Area must be checked in the Options dialog box, User Preferences tab.

Create Callout Blocks and Label Blocks (Advanced)

If you create a block to be used as a callout block or label block in a sheet set, you can use a placeholder field to display information such as view title or sheet number. The callout or label block must be defined in a DWG or DWT file that is specified in the Sheet Set Properties dialog box. Later, you can insert the callout or label block from a shortcut menu on the Sheet Views tab in the Sheet Set Manager.

For the field to display the correct information about a view or sheets on which you later insert it, the field must be included within a block attribute, not text, when you define the block. To create the block attribute definition, insert a placeholder field as the value, select the Preset option, and specify a tag.

NOTE If you create your own label blocks and callout blocks, set any attribute definitions to Preset to avoid prompts when placing these blocks in a drawing.

For more information about fields, see [Insert Fields](#).

Place a Sheet View (Advanced)

The Sheet Set Manager automates and enhances the process for adding views to a sheet. A view on a sheet, called a *sheet view*, consists of several coincident

entities: an xref or geometry in model space, a layout viewport on a sheet, and a named view in paper space.

- The sheet view can display model space from a different drawing file. In this case, that drawing is attached as an xref in your current drawing. The layers of that drawing file are displayed only in the sheet view that you create.

NOTE The xref is attached using a relative path. If you need to change the path to an absolute path, use the External References palette.

- A layout viewport that displays the model space view is created on your current sheet.
- A named view that encompasses the area of the layout viewport is created in paper space.

When you place a sheet view on a sheet, all the layers in the current drawing (including layer 0) are frozen in the new viewport created by the view. The layers are shown as frozen in the Current VP Freeze column of the Layer Properties Manager.

If you need to remove a sheet view from a sheet, you can delete the layout viewport to remove the view. However, to remove all unused items, you need to detach the xref and delete the named paper space view.

NOTE The easiest method for removing a sheet view immediately after placing it is to use UNDO.

See also:

["Create Multiple-View Drawing Layouts \(Paper Space\)" on page 209](#)

To import layouts from existing drawings as sheets

- 1 In the Sheet Set Manager, Sheet List tab, right-click the sheet set node, a subset node, or a sheet node. Click Import Layout as Sheet.
- 2 In the Import Layouts as Sheets dialog box, click Browse for Drawings and navigate to the drawing that you want to use.
To select several drawings, use SHIFT or CTRL when you click on the drawing files.
- 3 Click the check boxes of the layouts to be imported as sheets in the current sheet set. Click Import Checked.
- 4 (Optional) On the Sheet List tab, right-click a newly imported sheet. Click Rename & Renumber.

- 5 (Optional) In the Rename & Renumber Sheet dialog box, make any changes to the sheet number and sheet title. Click OK.

NOTE Changes to the sheet number and sheet title have no effect on the drawing file name unless a file rename option is checked.

Standard toolbar



Command line: SHEETSET

To create a new sheet

- 1 In the Sheet Set Manager, Sheet List tab, right-click either the sheet set node, a subset node, or a sheet node. Click New Sheet.
- 2 In the New Sheet dialog box, enter the sheet number and sheet title. Click OK.

The new sheet is created from the drawing template file specified in Sheet Set Properties for the default new sheet creation template.

NOTE You can change the default folder location for new sheets from the Sheet Set Properties or the Subset Properties dialog box.

Standard toolbar



Command line: SHEETSET

To open a drawing file within a sheet set

- 1 In the Sheet Set Manager, open a sheet set.
- 2 On the Sheet List tab, double-click a sheet. To select several drawings, use SHIFT or CTRL when you click on the drawing files.

The drawing file for the sheet is opened. Any changes made to the drawing that are relevant to the sheet list are updated in the sheet set data (DST) file.

Standard toolbar



Command line: SHEETSET

To open a drawing file read-only within a sheet set

- 1 In the Sheet Set Manager, open a sheet set.
- 2 On the Sheet List tab, right-click a sheet. Click Open Read-Only.

Standard toolbar



Command line: SHEETSET

To remove a sheet from a sheet set

- 1 In the Sheet Set Manager, open a sheet set.
- 2 On the Sheet List tab, right-click the sheet that you want to remove. Click Remove Sheet.

The specified sheet is removed from the sheet list. The sheet and the drawing file are not deleted, however. The sheet can be added to a different sheet set.

NOTE The DWG file associated with a sheet does not have to be accessible for the sheet to be removed from the sheet set.

Standard toolbar



Command line: SHEETSET

To reassociate a sheet in a sheet set

- 1 In the Sheet Set Manager, open a sheet set.
- 2 On the Sheet List tab, open the sheet that you want to reassociate.
- 3 In the Sheet Set Manager, right-click the sheet. Click Remove Sheet.
- 4 Save the drawing.

- 5 In the Sheet Set Manager, right-click the Sheet Set. Click Import Layout as Sheet.
- 6 In the Import Layout as Sheet dialog box, click Browse for Drawings. Navigate to the drawing that you want to use.
- 7 Click the check box of the layout to be reassociated as a sheet in the current sheet set. Click Import Checked.

Standard toolbar



Command line: SHEETSET

To add a view to a sheet

- 1 In the Sheet Set Manager, open a sheet set.
- 2 On the Sheet List tab, do *one* of the following:
 - Double-click a sheet to open it.
 - Create a new sheet and open it.
- 3 On the Model Views tab, click the plus sign (+) next to a folder to list the drawings in the folder.
- 4 From the list of drawing files, do *one* of the following:
 - To add a model space view to a sheet, click the plus sign (+) next to a drawing file to list its named model space views. Right-click a model space view.
 - To add an entire drawing as a view in a sheet, right-click a drawing file.
- 5 Click Place on Sheet.

NOTE As an alternative to the previous two steps, you can drag a model space view or a drawing from the Model Views tab to a sheet.

- 6 Right-click on the sheet. Click the scale for the sheet view.

- 7 Specify the insertion point for the sheet view.

The specified view is added to the sheet. If a view label block is defined in the sheet set properties, a view label that displays view-specific information is automatically placed on the sheet.

Standard toolbar



Command line: SHEETSET

To add a sheet list table

- 1 In the Sheet Set Manager, open a sheet set.
- 2 Right-click the sheet set name. Click Insert Sheet List Table.
- 3 In the Insert Sheet List Table dialog box, do the following:
 - Set the Table Style in the Table Style Settings group.
 - Specify Title Text for the table from the Table Data Settings group.
 - Add, remove, or change the order of the column entries.
- 4 Click OK to close the dialog box and add the table to the drawing.

Standard toolbar



Command line: SHEETSET

To edit a sheet list table

- 1 Select a cell in an existing sheet list table.
- 2 Right-click the cell. Click Edit Sheet List Table Settings.
- 3 In the Edit Sheet List Table Settings dialog box, do one or more of the following:
 - Change the Table Style if you don't like the current format.
 - Edit the Title Text for the table.
 - Add, remove, or change the order of the column entries.
 - Change the data type or heading text for column entries.
- 4 Click OK to close the dialog box and change the table to the drawing.

Shortcut menu: Select any cell in the sheet list table ► Right-click to open the shortcut menu ► Edit Sheet List Table Settings.

To update a sheet list table

- 1 Select a cell in an existing sheet list table.
- 2 Right-click the cell. Click Update Sheet List Table.

Shortcut menu: Select any cell in the sheet list table ➤ Right-click to open the shortcut menu ➤ Update Sheet List Table.

To add a block to be used for sheet view callouts

- 1 In the Sheet Set Manager, open a sheet set.
- 2 On the Sheet Views tab, right-click the sheet set node.
- 3 In the Sheet Set Properties dialog box, click Callout Blocks. Click the [...] button.
- 4 In the List of Blocks dialog box, do *one* of the following:
 - Click a block from the list of blocks.
 - Click the Add button and specify a new block to be added to the list.
- 5 Click OK to close each dialog box.

Standard toolbar



Command line: SHEETSET

To change the default block used for sheet view labels

- 1 In the Sheet Set Manager, open a sheet set.
- 2 On the Sheet Views tab, right-click the sheet set node.
- 3 In the Sheet Set Properties dialog box, click Label Block for Views. Click the [...] button.
- 4 In the Select Block dialog box, specify a new block to be used as the default view label block.
- 5 Click OK to close each dialog box.

Standard toolbar



Command line: SHEETSET

To add a callout block to a sheet

- 1 In the Sheet Set Manager, open a sheet set.
- 2 On the Sheet Views tab, right-click the view with which you want to associate a callout. Click Place Callout Block.
- 3 Specify the insertion point of the callout block.

The callout block is placed on the sheet. The callout block automatically displays view-specific information about the view with which it is associated.

Standard toolbar



Command line: SHEETSET

To create a sheet list table on a title sheet

- 1 In the Sheet Set Manager, open a sheet set.
- 2 On the Sheet List tab, double-click the sheet to be used as the title sheet.
- 3 Right-click the sheet set node. Click Insert Sheet List Table.
- 4 In the Sheet List Table dialog box, enter the title of the table. Make any formatting changes needed.
- 5 Click OK.
- 6 Specify the insertion point of the table.

The sheet list table automatically generates a list of all the sheets in the sheet set.

Standard toolbar



Command line: SHEETSET

Include Information with Sheets and Sheet Sets

Sheet sets, subsets, and sheets include several types of information. This information, called properties, includes titles, descriptions, file paths, and custom properties that you define.

Different Properties for Different Levels (Owners)

Sheet sets, subsets, and sheets represent different levels of organization, and each of these includes different types of properties. You specify the values for these properties when you create the sheet set, subset, or sheet.

In addition, you can define custom properties for a sheet and a sheet set. The values for custom properties for sheets are typically specific to each sheet. For example, a custom property for a sheet might include the name of the designer. The values for custom properties for a sheet set are typically specific to a project. For example, a custom property for a sheet set might include the contract number.

You cannot create custom properties for subsets.

View and Edit Properties

You can view and edit properties from the Sheet List tab by right-clicking the name of the sheet set, subset, or sheet. On the shortcut menu, click Properties. The properties and values that are displayed in the Properties dialog box depend on what you selected. You can edit the values of the properties by clicking a value.

To edit the properties of a sheet set, subset, sheet, or view category

- 1 In the Sheet Set Manager, open a sheet set.
- 2 Do *one* of the following:
 - On the Sheet List tab, right-click the sheet set node, a subset, or a sheet.
 - On the Sheet Views tab, right-click a view category.
- 3 Click Properties.
- 4 In the dialog box that is displayed, click any field and enter the changes.
- 5 Click OK.

Standard toolbar



Command line: SHEETSET

To create a custom property for a sheet set

- 1 In the Sheet Set Manager, open a sheet set.
- 2 On the Sheet List tab, right-click the sheet set node. Click Properties.
- 3 In the Sheet Set Properties dialog box, click the Edit Custom Properties button.
- 4 In the Custom Properties dialog box, do *one* of the following:
 - Click Add to create a new custom property for the sheet set. Click the name and the default value to enter the information for these fields. To assign this property to the sheet set, make sure that Sheet Set is selected under Owner.
 - Click the name and the default value to change the information for these fields.
 - Click a custom property from the list. Click Delete to remove the property.
- 5 Click OK to close each dialog box.

Standard toolbar



Command line: SHEETSET

To create a custom property for a sheet

- 1 In the Sheet Set Manager, open a sheet set.
- 2 On the Sheet List tab, right-click the sheet set node.
- 3 In the Sheet Set Properties dialog box, click the Edit Custom Properties button.
- 4 In the Custom Properties dialog box, do *one* of the following:
 - Click Add to create a new custom property for the sheet. Click the name and the default value to enter the information for these fields.

To assign this property to the sheet, make sure that Sheet is selected under Owner.

- Click the name and the default value to change the information for these fields. With this method, you can change the value for every sheet in the sheet set.
- Click a custom property from the list. Click Delete to remove the property.

5 Click OK to close each dialog box.

Standard toolbar



Command line: SHEETSET

Publish, Transmit, and Archive Sheet Sets

After you have organized drawings into a sheet set, you can publish, transmit, and archive the sheet set as a package.

- **Publish a sheet set.** Use the Publish feature to create single and multiple-sheet Design Web Format™ (DWF™) files from a sheet set or portion of a sheet set. Alternatively, you can plot the sheet set. For more information, see “Publish a Sheet Set” on page 1043. For information about DWF files, see “Plot DWF Files” on page 1019.
- **Set options for properties to include in a published DWF file.** You can decide what types of information to reveal in your published DWF files. The types of metadata that you can include are sheet and sheet set properties, block properties and attributes, dynamic block properties and attributes, and properties contained in custom objects. Metadata is only included when you publish to DWF; it is not available when plotting to DWF.
- **Transmit a sheet set.** Package and send a sheet set or a portion of a sheet set over the Internet. For more information, see “Package a Set of Files for Internet Transmission”.
- **Archive a sheet set.** Package a sheet set or portion of a sheet set for storage. This is very similar to packaging a transmittal set, except that you specify a folder for the archive and you do not transmit the package. For more information, see the ARCHIVE command.

Use Page Setups

Page setups provide the settings that are used for publishing and plotting. When you create a sheet set, you specify a drawing template (DWT) file that contains one or more page setups for all new sheets. This DWT file is called the *sheet creation template*.

Another DWT file, called the *page setup overrides* file, contains page setups that can be specified to override the page setups in each sheet. You specify the page setup overrides file in the Sheet Set Properties dialog box.

When you publish a sheet set, you can use the page setups defined in each drawing file, you can use the page setup overrides for all drawing files, or you can publish to a DWF file. For more information about page setups, see “Use Named Page Setups with Sheet Sets” on page 971.

Save a Selection of Sheets

You can select part of a sheet set for publishing and transmitting. On the Sheet List tab, you can select individual sheets using standard Microsoft® Windows® selection methods—press CTRL or SHIFT when you click items. You can specify all the sheets in a sheet subset by clicking the subset node.

You will likely need to perform operations on the same group of sheets in a sheet set repeatedly. To speed up selecting the sheets and to ensure that the same sheets are selected each time, you can use the Sheet Set Manager to reselect groups of sheets by name. These named groups of sheets are called *sheet selections*. You can create and manage sheet selections by using the Sheet Selections button at the top of the Sheet List tab.

To save a sheet selection

- 1 In the Sheet Set Manager, open a sheet set.
- 2 On the Sheet List tab, click the sheets and subsets to include in the sheet selection.
You can use CTRL or SHIFT to specify several items from the list.
- 3 In the Sheet List Manager, near the upper-right corner, click the Sheet Selections button. Click Create.
- 4 In the New Sheet Selection dialog box, enter the name of the sheet selection. Click OK.

Sheet Set Manager window



Command line: SHEETSET

To restore a sheet selection

- 1 In the Sheet Set Manager, open a sheet set.
- 2 On the Sheet List tab, near the upper-right corner, click the Sheet Selections button. Select the name of the sheet selection that you want to restore.
The sheet selection is activated and available for publishing and transmitting operations.

Sheet Set Manager window



Command line: SHEETSET

To rename or delete a sheet selection

- 1 In the Sheet Set Manager, open a sheet set.
- 2 On the Sheet List tab, near the upper-right corner, click the Sheet Selections button. Click Manage.
- 3 In the Sheet Selections dialog box, select the name of a sheet selection and do *one* of the following:
 - Click Rename to rename the sheet selection. Enter the new name for the sheet selection.
 - Click Delete to remove the sheet selection name from the list. Click Yes to confirm that you want to remove this sheet selection name.
- 4 Click OK.

Sheet Set Manager window



Command line: SHEETSET

To modify an existing sheet selection

NOTE You cannot modify a sheet selection directly. Instead, you activate a sheet selection and modify it, delete the name of the sheet selection, and then create a new sheet selection in its place.

- 1 In the Sheet Set Manager, open a sheet set.
- 2 On the Sheet List tab, near the upper-right corner, click the Sheet Selections button. Select the name of the sheet selection to modify.
- 3 Use the CTRL key to add and remove items from the sheet selection.
- 4 On the Sheet List Manager, near the upper-right corner, click the Sheet Selections button. Click Manage.
- 5 Click Delete to remove the sheet selection name from the list. Click Yes to confirm that you want to remove this sheet selection name.
- 6 Click OK to close the Sheet Selections dialog box.
- 7 On the Sheet List Manager, near the upper-right corner, click the Sheet Selections button. Click Create.
- 8 In the New Sheet Selection dialog box, enter the name of the sheet selection. Click OK.

Sheet Set Manager window



Command line: SHEETSET

To include layer, block, sheet set, and sheet information in a published DWF file

- 1 In the Sheet Set Manager, Sheet List tab, select the sheet set to publish in a DWF file.
- 2 Click the Publish button. Click Sheet Set Publish Options.
- 3 In the Sheet Set Publish Options dialog box, under DWF Data Options, depending on what you want to include in the published DWF file, click any of the following to change the option to “Include.”
 - Layer information
 - Sheet set information (attributes you can choose to include are description and custom properties)

- Sheet information (attributes you can choose to include are sheet title, sheet number, description, sheet set, subset, and sheet custom properties)

- Block information

NOTE Some sheet properties are always published in the DWF file (no matter what setting you make in the Sheet Set Publish Options dialog box). These are: sheet name, sheet size, author, creator, creation time, and modification time.

- 4 Click OK.

Sheet Set Manager window



Command line: SHEETSET

To use a block template file to include block properties and attributes in a published DWF file

- 1 In the Sheet Set Manager, Sheet List tab, select the sheet set to publish in a DWF file.
- 2 Click the Publish button. Click Sheet Set Publish Options.
- 3 In the Sheet Set Publish Options dialog box, under DWF Data Options, Block Information, click to display the drop-down list, and select “Include.”
- 4 Under DWF Data Options, Block Template File, select the block template file you want to use or create a new one. Click OK.

The block template file defining the blocks and their properties and attributes will be included in the published DWF file.

Sheet Set Manager window



Command line: SHEETSET

Use Sheet Sets in a Team

You can use sheet sets in a team that can involve network access, Internet collaboration, and email transmittal. The team can also include people who use software that does not include Sheet Set Manager.

Work in a Team That Uses Sheet Set Manager

When you use sheet sets in a team, each member should have network access to the sheet set data (DST) file and the drawing template (DWT) files associated with the sheet set. Each team member can open the sheet set to load the sheet set information from the DST file into the Sheet Set Manager.

Any changes that any team member makes opens the DST file briefly and updates the information stored in that file. When the DST file is opened, a lock icon is displayed next to the sheet set name in the top left corner of the Sheet Set Manager.



A green dot in the lock icon indicates that the Sheet Set Manager session on your computer has temporarily locked the DST file.



A red dot indicates that the Sheet Set Manager session on a team member's computer has temporarily locked the DST file.



A yellow dot in the lock icon means that the sheet is in a special state; for example, its file properties may be set to Read-Only.

Other members of the team can automatically see changes to the sheet set in the Sheet Set Manager tree view.

If each member of the team has access to the sheet set DWT files, new drawing files and their sheets are created using the same drawing template file; page setups for these drawings are also standardized.

NOTE If two or more users access the same sheet files through different logical drives on a network, each will in turn be prompted to resave the sheet set using their own logical drive. To avoid unnecessary saving, users should map the same logical drives, if possible.

Status data for sheets in the current sheet set is also available to other team members. This status data is displayed in the tree view and indicates one of the following conditions:



The sheet is available for editing.



The sheet is locked.



The sheet is missing or found in an unexpected folder location.

The active sheets of other team members are automatically polled for status changes; the tree view is updated in your session of the Sheet Set Manager. The polling cycle skips the poll interval in your session when a command is active. To force a sheet status update, click Refresh Sheet Status on the Sheet List tab.

You can click any sheet to display more information in the Details area of the Sheet Set Manager.

NOTE A false lock icon may be displayed if there is a network problem or if the program terminates unexpectedly. If you suspect a problem, click the sheet to display more information.

Work in a Team That Does Not Use Sheet Set Manager

With some limitations, you can use sheet sets in a team with members who do not have network access, or do not have access to the Sheet Set Manager. These team members may be using AutoCAD LT or an older version of AutoCAD. In those circumstances, not all members of the team will have access to the DST file. However, relevant information from the DST file is stored (cached) in each drawing file, and sheet set information, such as custom properties, is preserved when the drawing file is shared by other team members.

After a member of the team changes information in the DST file, the information in several drawing files might need to be updated. With the sheet set open, update a sheet by opening and saving the sheet.

You can update all sheets in a sheet set automatically with the Resave All Sheets option in the sheet set shortcut menu. Drawing files saved in a previous DWG file format are saved without changing the format.

NOTE In a network environment, make sure that all drawing files used in the current sheet set that are opened by other users are closed before performing the Resave All Sheets operation.

To resave all sheets in a sheet set

- 1 In the Sheet Set Manager, open a sheet set.
- 2 On the Sheet List tab, right-click the sheet set node. Click Resave All Sheets.

All sheets in the sheet set are resaved. This updates the sheet set information that is stored in each drawing file.

Standard toolbar



Command line: SHEETSET

Part 5

Create and Modify Objects

- Chapter 17 Control the Properties of Objects
- Chapter 18 Use Precision Tools
- Chapter 19 Draw Geometric Objects
- Chapter 20 Create and Use Blocks (Symbols)
- Chapter 21 Change Existing Objects

Control the Properties of Objects

17

In this chapter

- Overview of Object Properties
- Display and Change the Properties of Objects
- Copy Properties Between Objects
- Work with Layers
- Work with Colors
- Work with Linetypes
- Control Lineweights
- Control the Display Properties of Certain Objects

You can organize objects in your drawing and control how they are displayed and plotted by changing their properties, which include layer, linetype, color, linewidth, and plot style.

Layers are like transparent overlays on which you organize and group different kinds of drawing information. The objects you create have common properties including colors, linetypes, and linewidths. An object can assume these properties from the layer it is drawn on, or properties can be specifically assigned to individual objects. Color helps you distinguish similar elements in your drawings, while linetypes help you differentiate easily between different drafting elements, such as centerlines or hidden lines. Lineweights represent the size or type of an object through width, enhancing your drawing and increasing legibility. Organizing layers and the objects on layers makes it easier to manage the information in your drawings.

Overview of Object Properties

Every object you draw has properties. Some properties are general and apply to most objects; for example, layer, color, linetype, and plot style. Other properties are object-specific; for example, the properties of a circle include radius and area, and the properties of a line include length and angle.

Most general properties can be assigned to an object by layer or can be assigned to an object directly.

- When a property is set to the value BYLAYER, the object is assigned the same value as the layer on which it is drawn.

For example, if a line drawn on Layer0 is assigned the color BYLAYER, and Layer0 is assigned the color Red, the line is red.

- When a property is set to a specific value, that value overrides the value set for the layer.

For example, if a line drawn on Layer0 is assigned the color Blue, and Layer0 is assigned the color Red, the line is blue.

See also:

[Control the Color and Linetype Properties in Blocks](#)

Display and Change the Properties of Objects

You can display and change the current properties for any object in your drawing in the following ways:

- Open the Properties palette and view and change the settings for all properties of the object.
- View and change the settings in the Layer control on the Layers toolbar and the Color, Linetype, Lineweight, and Plot Style controls on the Properties toolbar.
- Use LIST to view information in the text window.
- Use the ID command to display a coordinate location.

Use the Properties Palette

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

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

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

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

See also:

- [Control the Color and Linetype Properties in Blocks](#)
- [Filter Selection Sets](#)
- [Set Interface Options](#)

To remove the description pane at the bottom of the Properties palette

- 1 On the Properties palette, right-click the title bar.
- 2 On the shortcut menu, click Description to clear the check mark.

To display the properties of a single object

- 1 Select the object.
- 2 Right-click in the drawing. Click Properties.
The Properties palette displays the properties of the selected object.
Alternatively, you can double-click almost any object to display the Properties palette.

Standard toolbar



Command line: PROPERTIES

To list database information for several objects

- 1 Click Tools menu ► Inquiry ► List.
- 2 Select one or more objects and press ENTER.
The text window displays a report.

Inquiry toolbar



Command line: LIST

To display the coordinate values of a point

- 1 Click Tools menu ► Inquiry ► ID Point.
- 2 Specify the point whose coordinate values you want to identify.
The *X*, *Y*, and *Z* values are displayed on the command line.

Inquiry toolbar



Command line: ID

To change properties of objects in the Properties palette

- 1 Select one or more objects.
- 2 Right-click in the drawing. Click Properties.
- 3 In the Properties palette, use the scroll bar next to the title bar to scroll through the list of properties. You can click the arrow at the right of each category to expand or collapse the list.
- 4 Select the value you want to change; use one of the following methods to change the value:
 - Enter a new value.
 - Click the down arrow at the right and select a value from the list.
 - Click the Pick Point button to use the pointing device to change a coordinate value.

- Click the QuickCalc calculator button to calculate a new value.
- Click the left or right arrow to increase or decrease the value.
- Click the [...] button and change the property value in a dialog box.

Changes are applied immediately.

- To undo a change, right-click an empty area in the Properties palette. Click Undo.
- Press ESC to remove the selection.

Standard toolbar



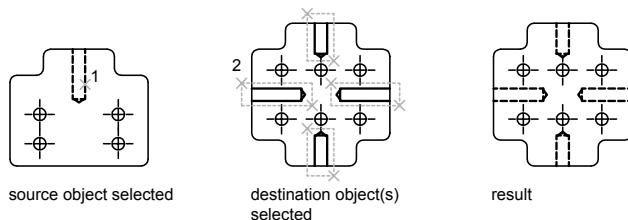
Command line: PROPERTIES

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, and 3D thickness.

By default, all applicable properties are automatically copied from the first object you selected to the other objects. If you don't want a specific property or properties to be copied, use the Settings option to suppress the copying of that property. You can choose the Settings option at any time during the command.



To copy properties from one object to other objects

- 1 Click Standard Toolbar ► Match Properties.

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

Standard toolbar



Command line: MATCHPROP

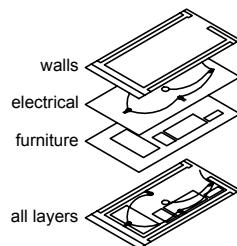
Work with Layers

Layers are like transparent overlays on which you organize and group different kinds of drawing information.

Overview of Layers

Layers are used to group information in a drawing by function and to enforce linetype, color, and other standards.

Layers are the equivalent of the overlays used in paper-based drafting. Layers are the primary organizational tool used in drawing. You use layers to group information by function and to enforce linetype, color, and other standards.



By creating layers, you can associate similar types of objects by assigning them to the same layer. For example, you can put construction lines, text, dimensions, and title blocks on separate layers. You can then control

- Whether objects on a layer are visible in any viewports
- Whether and how objects are plotted
- What color is assigned to all objects on a layer
- What default linetype and linewidth are assigned to all objects on a layer
- Whether objects on a layer can be modified

Every drawing includes a layer named 0. Layer 0 cannot be deleted or renamed. It has two purposes:

- Ensure that every drawing includes at least one layer
- Provide a special layer that relates to controlling colors in blocks

NOTE It is recommended that you create several new layers with which to organize your drawing rather than create your entire drawing on layer 0.

Use Layers to Manage Complexity

You can use layers to control the visibility of objects and to assign properties to objects. Layers can be locked to prevent objects from being modified.

You can reduce the visual complexity of a drawing and improve display performance by controlling which objects are displayed or plotted. For example, you can use layers to control the properties and visibility of similar objects, such as electrical parts or dimensions. Also, you can lock a layer to prevent objects on that layer from being accidentally selected and modified.

Control the Visibility of Objects on a Layer

You can make drawing layers invisible either by turning them off or by freezing them. Turning off or freezing layers is useful if you need an unobstructed view when working in detail on a particular layer or set of layers or if you don't want to plot details such as reference lines. Whether you choose to freeze layers or turn them off depends on how you work and on the size of your drawing.

- **On/Off**. Objects on turned-off layers are invisible, but they still hide objects when you use HIDE. When you switch layers on and off, the drawing is not regenerated.

■ **Freeze/Thaw.** Objects on frozen layers are invisible and do not hide other objects. Thawing one or more layers causes the drawing to be regenerated. Freezing and thawing layers takes more time than turning layers on and off.

In a layout (paper space), you can make some layers visible only in certain viewports.

Assign a Default Color and Linetype to a Layer

Each layer has associated properties such as color and linetype that are assumed by all objects on that layer. For example, if the Color control on the Properties toolbar is set to BYLAYER, the color of new objects is determined by the color setting for the layer in the Layer Properties Manager.

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

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

Control Whether Objects on a Layer Can Be Modified

When a layer is locked, none of the objects on that layer can be modified until you unlock the layer. Locking layers reduces the possibility of modifying objects accidentally. You can still apply object snaps to objects on a locked layer and perform other operations that do not modify those objects.

To make a selected layer the current layer

- 1 Click Format menu ▶ Layer Tools ▶ Make Objects Layer Current.
- 2 Select an object on the layer you want to make current.

Layers toolbar



Command line: LAYMCUR

To copy an object to another layer

- 1 Click Format menu ▶ Layer Tools ▶ Copy Objects to New Layer.
- 2 Select the objects you want to copy.
- 3 Press ENTER.

- 4 Select an object on the layer where you want the copied object to be placed.
- 5 Do one of the following:
 - Specify a base point, and then specify a second point for the location of the object on the destination layer.
 - Enter the displacement in the form of a Cartesian, polar, cylindrical, or spherical coordinate value. At the prompt for the second point of displacement, press ENTER.

Layers II toolbar



Command line: COPYTOLAYER

To dynamically display layers

- 1 Click Format menu ▶ Layer Tools ▶ Layer Walk.
- 2 In the LayerWalk dialog box, click a layer to view the objects on that layer. To select more than one layer, press CTRL and click the layers you want to view.
The drawing displays the objects on the layers you selected.
- 3 Click Close.

Layers II toolbar



Command line: LAYWALK

To filter a layer list

- 1 Click Format menu ▶ Layer Tools ▶ Layer Walk.
- 2 In the LayerWalk dialog box, Filter list, do one of the following:
 - Enter an existing filter name or select a filter from the Filter drop-down list.
 - Enter filter criteria (using wild-card characters) to create a list of the layers you want filtered. For example, if want to filter layers named "0" through "3", you could enter a filter criteria of [0-3]*.

NOTE For more information about wild-card characters, see the “Wild-Card Characters” section of Filter and Sort the List of Layers.

The layer list displays layers defined in the filter.

- 3 Click Close.

Layers II toolbar



Command line: LAYWALK

To create and save a layer filter in the LayerWalk dialog box

- 1 Click Format menu ► Layer Tools ► Layer Walk.
- 2 In the LayerWalk dialog box, Filter list, enter filter criteria (using wild-card characters) to create a list of the layers you want filtered. For example, if want to filter layers “0” through “3”, enter a filter criteria of [0-3]*.

NOTE For more information about wild-card characters, see the “Wild-Card Characters” section of Filter and Sort the List of Layers.

- 3 Right-click, and then click Save Current Filter.
- 4 Click Close.

Layers II toolbar



Command line: LAYWALK

To turn on or off an active layer filter

- 1 Click Format menu ► Layer Tools ► Layer Walk.
- 2 In the LayerWalk dialog box, Filter list, enter an existing filter name or select a filter from the Filter drop-down list.
- 3 Click the Filter check box to display or hide the layers in the layer list that are defined in the active filter. The display of the layers in the drawing itself does not change.
- 4 Click Close.

Layers II toolbar



Command line: LAYWALK

To turn the Always Show option on or off in the LayerWalk dialog box

- 1 Click Format menu ► Layer Tools ► Layer Walk.
- 2 In the LayerWalk dialog box, Layer list, double-click the layer you want to set to Always Show. To select more than one layer, press SHIFT and double-click the additional layers.
An asterisk (*) is displayed to the left of each layer that you set to Always Show. These layers are displayed in the drawing regardless of whether or not they are in an active filter.
- 3 Click Close.

Layers II toolbar



Command line: LAYWALK

To purge unreferenced layers using the LayerWalk dialog box

- 1 Open a drawing that contains one or more unreferenced layers that you want to purge.
- 2 Click Format menu ► Layer Tools ► Layer Walk.
- 3 In the LayerWalk dialog box, right-click. Click Select Unreferenced.

NOTE The Purge button is active only when the layers you select are unreferenced.

- 4 Click Purge.
- 5 Click Close.

NOTE You can also purge layers in the Layer Properties Manager. See To remove an unused layer

Command line: LAYWALK

Layers II toolbar



To move an object to the current layer

- 1 Click Format menu ► Layer Tools ► Change to Current Layer.
- 2 Select the objects you want to move to the current layer.
- 3 Press ENTER.

Layers II toolbar



Command line: LAYCUR

To delete a layer by selecting an object on that layer

- 1 Click Format menu ► Layer Tools ► Layer Delete.
- 2 Select an object on the layer you want to delete.
- 3 Press ENTER.

All objects on the selected layer and the layer itself are deleted.

Command line: LAYDEL

To freeze a layer by selecting an object on that layer

- 1 Click Format menu ► Layer Tools ► Layer Freeze.
- 2 Select an object on the layer you want to freeze.,.
- 3 Press ENTER.

The selected layer is frozen.

Layers II toolbar



Command line: LAYFRZ

To thaw all layers

- Click Format menu ► Layer Tools ► Thaw All Layers.

All layers (except layers that are frozen in a viewport) are thawed.

Command line: LAYTHW

To display selected layers while turning off all other layers

- 1 Click Format menu ► Layer Tools ► Layer Isolate.

- 2 Select an object on the layer you want to isolate.

- 3 Press ENTER.

The selected layer is isolated.

Layers II toolbar



Command line: LAYISO

To restore layer settings when exiting the LayerWalk dialog box

- 1 Click Format menu ► Layer Tools ► Layer Walk.

- 2 In the LayerWalk dialog box, click a layer to view the objects on that layer. To select more than one layer, click a layer in the list and drag the cursor over the layers you want to view.

- 3 Do one of the following:

- To return to the original view when you close the dialog box, make sure the Restore on Exit check box is selected.

- To return to the drawing with only the selected layers, make sure the Restore on Exit check box is blank.

- 4 Click Close.

Layers II toolbar



Command line: LAYWALK

To restore a drawing's layer states after isolating a layer

Layers II toolbar



Command line: LAYUNISO

To isolate an object's layer to the current viewport

- 1 Click a layout tab.
 - 2 Click Format menu ► Layer Tools ► Isolate Layer to Current Viewport.
 - 3 Select the object whose layer you want to isolate. Press ENTER.
- The selected layer is frozen in all viewports except the current viewport.

Command line: LAYVPI

To turn a layer on and off by using the Layers toolbar

- 1 On the Layers toolbar, click the Layer Properties Manager control.
 - 2 In the Layer Properties Manager, click the light bulb for the layer name that you want to turn on or off.
- If the light bulb is yellow, the layer is turned on.

Layers toolbar



Command line: LAYER

To turn all layers on

Command line: LAYON

To turn a layer off by selecting an object on the layer

- 1 Click Format menu ► Layer Tools ► Layer Off
- 2 Select the object whose layer you want to turn off. Press ENTER.

Layers II toolbar



Command line: LAYOFF

To copy properties from one layer to other layers

- 1 Click Format menu ► Layer Tools ► Layer Match.
- 2 Select the object whose layer you want to change. Press ENTER.
- 3 Select an object on the layer where you want the object to move.
The object is moved to the selected layer.

Layers II toolbar



Command line: LAYMCH

To merge selected layers onto a destination layer

- 1 Click Format menu ► Layer Tools ► Layer Merge.
- 2 Select any object on the layer you want to delete. Press ENTER.
- 3 Select any object on the layer where you want all the objects from the first layer to be merged.
The objects from the first layer are moved to the selected layer, and the first layer is deleted.

Command line: LAYMRG

To assign a color to a layer

- 1 Click Format menu ► Layer.
- 2 In the Layer Properties Manager, select a layer. Click the color icon.
- 3 In the Select Color dialog box, select a color.
- 4 Click Apply to save your changes, or click OK to save and close.

Layers toolbar



Command line: LAYER

To change the properties of more than one layer

- 1 Click Format menu ► Layer.
- 2 In the Layer Properties Manager list view, use one of the following methods to select the layers:
 - Hold down CTRL and select layer names.
 - Right-click. Click Show Filters in Layer List to display a check mark, and then select a layer filter.
- 3 Click the icons for the properties you want to change.
- 4 Click Apply to save your changes, or click OK to save and close.

Layers toolbar



Command line: LAYER

To hide the tree view in the Layer Properties Manager

- Right-click in the list view. Clear the Show Filter Tree option.

To hide a column in the Layer Properties Manager

- Drag the right column separator to the left until the column is hidden.

To show the column, drag the column separator to the right.

Create and Name Layers

By grouping objects into layers, you can control their display and make changes quickly and efficiently.

You can create and name a layer for each conceptual grouping (such as walls or dimensions) and assign common properties to those layers. By grouping objects into layers, you can control their display and make changes quickly and efficiently.

The number of layers you can create in a drawing and the number of objects you can create per layer are virtually unlimited. Layers are usually dedicated to each conceptual grouping (such as walls or dimensions). Layers can have

alphanumeric names up to 255 characters long. In many cases, the layer names you choose are dictated by corporate, industry, or client standards.

The Layer Properties Manager sorts layers alphabetically by name. If you're organizing your own layer scheme, name layers carefully. By using common prefixes to name layers with related drawing components, you can use wild-card characters in layer name filters when you need to find those layers quickly.

NOTE If you consistently use a specific layering scheme, you can set up a drawing template with layers, linetypes, and colors already assigned. For more information about creating templates, see "Use a Template File to Start a Drawing" on page 111.

Copy Layers from Another Drawing

You can use DesignCenter™ to copy layers from any drawing to another by dragging. For example, you might have a drawing that contains all the standard layers needed for a project. You can create a new drawing and use DesignCenter to drag the predefined layers to the new drawing, which saves you time and ensures consistency between drawings.

You can also drag layers or copy layers by double-clicking or by clicking Insert on the shortcut menu.

NOTE You need to resolve duplicate layer names before you drag layers from DesignCenter.

Select a Layer to Draw On

As you draw, newly created objects are placed on the current layer. The current layer may be the default layer (0) or a layer you create and name yourself. You switch from one layer to another by making a different layer current; any subsequent objects you create are associated with the new current layer and use its color, linetype, and other properties. You cannot make a layer the current layer if it is frozen or if it is an xref-dependent layer.

Remove Layers

You can remove unused layers from your drawing with PURGE or by deleting the layer from the Layer Properties Manager. You can delete only unreferenced layers. Referenced layers include layers 0 and DEFPOINTS, layers containing objects (including objects in block definitions), the current layer, and xref-dependent layers.

WARNING Be careful about deleting layers if you are working on a drawing in a shared project or one based on a set of layering standards.

To create a new layer

- 1 Click Format menu ► Layer.
- 2 In the Layer Properties Manager, click the New Layer button.
A layer name, such as LAYER1, is automatically added to the layer list.
- 3 Enter a new layer name by typing over the highlighted layer name.
A layer name can include up to 255 characters: letters, digits, and the special characters dollar sign (\$), hyphen (-), and underscore (_). Use a reverse quote (`) before other special characters so that the characters are not interpreted as wild-card characters. Layer names cannot include blank spaces.
- 4 To change the properties, click icons.
When you click Color, Linetype, Lineweight, or Plot Style, a dialog box is displayed.
- 5 (Optional) Click in the Description column and enter text.
- 6 Click Apply to save your changes, or click OK to save and close.

Layers toolbar



Command line: LAYER

To remove an unused layer

- 1 On the Layers toolbar, click Layer.
- 2 In the Layer Properties Manager, select the layer. Click the Delete Layer button.
Layers that have objects assigned to them cannot be removed until those objects are reassigned to a different layer or are deleted. Layers 0 and DEFPOINTS and the current layer cannot be removed.
- 3 Click Apply to save your changes, or click OK to save and close.
The selected layer is removed.

Layers toolbar



Command line: LAYER

To purge all unused layers

- 1 Click File menu ► Drawing Utilities ► Purge.

The Purge dialog box displays a tree view of object types with items that can be purged (removed from the drawing).

- 2 To purge unreferenced layers, use one of the following methods:
 - To purge all unreferenced layers, select Layers.
 - To purge specific layers, double-click Layers to expand the tree view. Select the layers to be purged.
- 3 If the item you want to purge is not listed, select View Items You Cannot Purge, select the layer, and read the explanation.
- 4 You are prompted to confirm each item in the list. If you do not want to confirm each purge, clear the Confirm Each Item to Be Purged option.
- 5 Click Purge.
- 6 To confirm the purging of each item, respond to the prompt by choosing Yes or No, or Yes to All if more than one item is selected.
- 7 Click Close.

Command line: PURGE

Change Layer Settings and Layer Properties

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

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

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

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

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

Undo Changes to Layer Settings

You can use Layer Previous to undo changes you make to layer settings. For example, if you freeze several layers and change some of the geometry in a drawing, and then want to thaw the frozen layers, you can do this with a single command without affecting the geometry changes. In another example, if you changed the color and linetype of several layers but later decide you prefer the old properties, you can use Layer Previous to undo the changes and restore the original layer settings.

When you use Layer Previous, it undoes the most recent change or set of changes made using either the Layer control or the Layer Properties Manager. Every change you make to layer settings is tracked and can be undone with Layer Previous. You can use LAYERPMODE to suspend layer property tracking when you don't need it, such as when you run large scripts. There is a modest performance gain in turning off Layer Previous tracking.

Layer Previous does not undo the following changes:

- **Renamed layers.** If you rename a layer and change its properties, Layer Previous restores the original properties but not the original layer name.
- **Deleted layers.** If you delete or purge a layer, using Layer Previous does not restore it.
- **Added layers.** If you add a new layer to a drawing, using Layer Previous does not remove it.

To change the layer of one or more objects

- 1 Select the objects whose layer you want to change.
- 2 On the Layers toolbar, click the Layer control.
- 3 Select the layer that you want to assign to the objects.
- 4 Press ESC to remove selection.

Layers toolbar



To change the default linetype assigned to a layer

- 1 Click Format menu ► Layer.
- 2 In the Layer Properties Manager, select the linetype for the layer you want to change.
- 3 In the Select Linetype dialog box, if the linetype you need is not displayed, click Load and use one of the following methods:
 - In the Load or Reload Linetypes dialog box, select one or more linetypes to load. Click OK to return to the Select Linetype dialog box.
 - In the Load or Reload Linetypes dialog box, click File to open additional linetype definition (LIN) files. Select one or more linetypes to load. Click OK to return to the Select Linetype dialog box.
- 4 Select the linetype you want to use. Click OK to return to the Layer Properties Manager.
- 5 Click Apply to save your changes, or click OK to save and close.

Layers toolbar



Command line: LAYER

To rename a layer

- 1 Click Format menu ► Layer.
- 2 In the Layer Properties Manager, select a layer. Click the name or press F2.
- 3 Enter a new name.
- 4 Click Apply to save your changes, or click OK to save and close.

Layers toolbar



Command line: LAYER

To rename more than one layer

- 1 On the command line, enter **rename**.

- 2 In the Rename dialog box, in the Named Objects list, select Layers.
 - 3 In Old Name, enter the old name, using wild-card characters; for example, enter **stairs\$***.
 - 4 In Rename To, enter the new name using wild-card characters; for example, enter **s_***.
- Results for this example are as follows: the layers STAIR\$LEVEL-1, STAIR\$LEVEL-2, STAIR\$LEVEL-3 are renamed S_LEVEL-1, S_LEVEL-2, S_LEVEL-3.
- 5 Click Rename To to apply changes and continue, or click OK.

To undo changes to layer settings

- On the Layers toolbar, click Layer Previous.

The command line displays the message “Restored previous layer status.”

NOTE LAYERP does not restore deleted or purged layers and does not remove a layer that was added. If you rename a layer and change its properties, Layer Previous restores the original properties but not the original layer name.

Layers toolbar



Command line: LAYERP

To set Layer Previous tracking on or off

- 1 At the Command prompt, enter **layerpmode**.
The current Layer Previous tracking status is displayed.
- 2 Enter **on** to turn on Layer Previous tracking of layer settings, or enter **off** to turn off tracking.

Command line: LAYERPMODE

Filter and Sort the List of Layers

You can control which layer names are listed in the Layer Properties Manager and sort them by name or by property, such as color or visibility.

A layer filter limits the display of layer names in the Layer Properties Manager and in the Layer control on the Layers toolbar. In a large drawing, you can use layer filters to display only the layers you need to work with.

There are two kinds of layer filters:

- **Layer property filters:** Includes layers that have names or other properties in common. For example, you can define a filter that includes all layers that are red and whose names include the letters *mech*.
- **Layer group filters:** Includes the layers that are put into the filter when you define it, regardless of their names or properties.

The tree view in the Layer Properties Manager displays default layer filters and any named filters that you create and save in the current drawing. The icon next to a layer filter indicates the type of filter. Three default filters are displayed:

- **All.** Displays all the layers in the current drawing.
- **All Used.** Displays all the layers on which objects in the current drawing are drawn.
- **Xref.** If xrefs are attached to the drawing, displays all the layers being referenced from other drawings.

Once you have named and defined a layer filter, you can select it in the tree view to display the layers in the list view. You can also apply the filter to the Layers toolbar, so that the Layer control displays only the layers in the current filter.

When you select a filter in the tree view and right-click, options on the shortcut menu can be used to delete, rename, or modify filters. For example, you can convert a layer property filter to a layer group filter. You can also change a property of all layers in a filter. The Isolate Group option turns off all layers in the drawing that are not in the selected filter.

Define a Layer Property Filter

A layer property filter is defined in the Layer Filter Properties dialog box, where you select any of the following properties you want to include in the filter definition:

- Layer names, colors, linetypes, linewidths, and plot styles
- Whether layers are in use
- Whether layers are turned on or off

- Whether layers are frozen or thawed in the current viewport or all viewports
- Whether layers are locked or unlocked
- Whether layers are set to be plotted

You use wild-card characters to filter layers by name. For example, if you want to display only layers that start with the letters *mech*, you can enter **mech***. See “Wild-Card Characters” for a complete list.

The layers in a layer property filter may change as the properties of the layers change. For example, if you define a layer property filter named Site that includes all layers with the letters *site* in the name and a CONTINUOUS linetype, and then you change the linetype of some of those layers, the layers with the new linetype are no longer part of the Site filter and are not displayed when you apply that filter.

Layer property filters can be nested under other properties filters or under group filters.

Define a Layer Group Filter

A layer group filter includes only those layers that you explicitly assign to it. If the properties of the layers assigned to the filter change, the layers are still part of the filter. Layer group filters can be nested only under other layer group filters.

Invert a Layer Filter

You can also invert a layer filter. For example, if all the site plan information in a drawing is contained in multiple layers that include the word *site* as part of the layer name, you can display all information except site plan information by first creating a filter definition that filters layers by name (**site**) and then using the Invert Filter option.

Sort Layers

Once you have created layers, you can sort them by name or other properties. In the Layer Properties Manager, click the column heading to sort layers by the property in that column. Layer names can be sorted in ascending or descending alphabetical order.

Wild-Card Characters

You can use wild-card characters to sort layers by name.

Character	Definition
# (Pound)	Matches any numeric digit

Character	Definition
@ (At)	Matches any alphabetic character
. (Period)	Matches any nonalphanumeric character
* (Asterisk)	Matches any string and can be used anywhere in the search string
? (Question mark)	Matches any single character; for example, ?BC matches ABC, 3BC, and so on
~ (Tilde)	Matches anything but the pattern; for example, ~*AB* matches all strings that don't contain AB
[]	Matches any one of the characters enclosed; for example, [AB]C matches AC and BC
[~]	Matches any character not enclosed; for example, [-AB]C matches XC but not AC
[-]	Specifies a range for a single character; for example, [A-G]C matches AC, BC, and so on to GC, but not HC
` (Reverse quote)	Reads the next character literally; for example, `~AB matches ~AB

NOTE If you want to use a wild-card character in the name of a named object, precede the character with a reverse quote (`) so that it is not interpreted as a wild-card character.

To quickly filter the display of layers by name

- 1 Click Format menu ► Layer.
- 2 In the Layer Properties Manager, click in Search for Layer below the tree view.
- 3 (Optional) To limit your search, select a layer filter in the tree view.
- 4 Enter a string of characters, including wild-card characters.

The list view displays all layers whose names match the string. For example, if you enter ***mech***, all layers with the letters *mech* in their names are displayed.

This quick filter is discarded when the Layer Properties Manager closes.

To filter the display of layers by layer property

- 1 Click Format menu ► Layer.

- 2** In the Layer Properties Manager, click the New Property Filter button.
- 3** In the Layer Filter Properties dialog box, enter a name for the filter.
- 4** Under Filter Definition, set the layer properties that you want to use to define the filter.
 - To filter by name, use wild-card characters.
 - To filter by property, click in the column for the property you want. Some properties display a dialog box when you click the [...] button.
 - To select more than one value for a property, right-click the row in the filter definition. Click Duplicate Row. Select another value for that property in the next row.

For example, the definition for a filter that displays only layers that are on and are either yellow or red has two rows. The first row of the filter definition has the On icon and red. The second row has the On icon and yellow.
- 5** Click Apply to save your changes, or click OK to save and close.

Layers toolbar



Command line: LAYER

To filter the display of layer names by selecting layers

- 1** On the Layers toolbar, click Layer Properties Manager.
- 2** In the Layer Properties Manager, click the New Group Filter button. A new layer group filter named GROUP FILTER1 is created in the tree view.
- 3** Enter a name for the filter.
- 4** In the tree view, click All or one of the other nodes to display layers in the list view.
- 5** In the list view, select the layers you want to add to the filter, and drag them to the filter name in the tree view.
- 6** Click Apply to save your changes, or click OK to save and close.

Layers toolbar



Command line: LAYER

To nest a layer filter under another layer filter

- 1 On the Layers toolbar, click Layer Properties Manager.
- 2 In the Layer Properties Manager tree view, select a layer filter.
 - A new layer property filter can be nested under a group filter or another property filter.
 - A new layer group filter can be nested only under another group filter.
- 3 Right-click. Click New Properties Filter or New Group Filter.
- 4 Use one of the following methods:
 - For a new property filter, the Layer Properties Filter dialog box is displayed. Under Filter Definition, set the layer properties that you want to use to define the filter. Click OK.
 - For a new group filter, a filter is added to the tree view. Rename it, select the parent filter to display its layers in the list view, and drag layers from the list view to the new layer group filter.
- 5 Click Apply to save your changes, or click OK to save and close.

Layers toolbar



Command line: LAYER

To sort the layer list in the Layer Properties Manager

- Click any column heading.

To reverse the sorting order, click a second time.

Layers toolbar



Command line: LAYER

To remove a layer from a layer group filter

- 1 On the Layers toolbar, click Layer Properties Manager.
- 2 In the Layer Properties Manager tree view, select a group filter.
- 3 Select the layer you want to remove.
- 4 Right-click in the list view. Click Remove from Group Filter.

Layers toolbar



Command line: LAYER

Save and Restore Layer Settings

You can save the current layer settings in a drawing as a named layer state and restore them later. Saving layer settings is convenient if you need to return to particular settings for all layers during different stages in completing a drawing or for plotting.

Save Layer Settings

Layer settings include layer states, such as on or locked, and layer properties, such as color or linetype. In a named layer state, you can choose which layer states and layer properties you want to restore later. For example, you can choose to restore only the Frozen/Thawed setting of the layers in a drawing, ignoring all other settings. When you restore that named layer state, all settings remain as they are currently set except whether each layer is frozen or thawed.

Using the Layer States Manager, you can export named layer states to an LAS file for use in other drawings. The layer states of external references cannot be exported.

Restore Layer Settings

When you restore a named layer state, the layer settings (layer states and layer properties) that were specified when the layer state was saved are restored by default. Because all the layer settings are saved in the named layer state, you can specify different settings at the time you restore. All layer settings that are not selected to be restored remain unchanged.

In addition, the layer that was current when the named layer state was saved is made current. If that layer no longer exists, the current layer does not change.

In addition to restoring named layer states, you can import named layer states that were previously exported to LAS files.

To save layer settings in a named layer state

- 1 Click Format menu ► Layer.
- 2 In the Layer Properties Manager, click the Layer States Manager button.
- 3 In the Layer States Manager, click New.
- 4 In the New Layer State to Save dialog box, enter a name for the new layer state, or select a name from the list.
- 5 (Optional) Add a description.
- 6 Click OK.
- 7 In the Layer States Manager, select the layer settings to restore by default.
- 8 (Optional) Select the Turn Off New Layers Not Found in Layer State option.
When this option is selected and you restore a named layer state, the drawing looks the same way it did when the named layer state was saved.
- 9 Click Close to exit the Layer States Manager. Click OK to exit the Layer Properties Manager.

Layers toolbar



Command line: LAYER

To export saved layer settings

- 1 Click Format menu ► Layer.
- 2 In the Layer Properties Manager, click the Layer States Manager button.
- 3 In the Layer States Manager, select a named layer state. Click Export.
- 4 In the Export Layer State dialog box, enter a file name. Specify a location for the layer state (LAS) file.
The *.las* file name extension is added automatically.
- 5 Click Save to exit the dialog box.
- 6 Click Close to exit the Layer States Manager. Click OK to exit the Layer Properties Manager.

Layers toolbar



Command line: LAYER

To restore layer settings

- 1 Click Format menu ► Layer.
- 2 In the Layer Properties Manager, click the Layer States Manager button.
- 3 In the Layer States Manager, select a named layer state.
- 4 Select the settings to restore. Click Restore.
The Layer States Manager closes.
- 5 Click OK.

Layers toolbar



Command line: LAYER

To modify a named layer state

- 1 Click Format menu ► Layer.
- 2 In the Layer Properties Manager, click the Layer States Manager button.
- 3 In the Layer States Manager, select the named layer state that you want to edit, rename, or delete.
- 4 Use one of the following methods:
 - To rename the named layer state, click the name and enter a new name.
 - To edit the description, click the description and enter new text.
 - To delete the saved layer state, click Delete.
 - To change which layer settings to restore by default, select or clear layer settings.
- 5 Click Close to exit the Layer States Manager. Click OK to exit the Layer Properties Manager.

Layers toolbar



Command line: LAYER

To import saved layer settings from another drawing

- 1 Click Format menu ► Layer.
- 2 In the Layer Properties Manager, click the Layer States Manager button.
- 3 In the Layer States Manager, click Import.
- 4 In the Import Layer State dialog box, select a file name with an *.las* file name extension. Click Open.
- 5 Click Yes to restore the named layer state now. Click No to add it to the Layer States Manager without restoring it.
If you restore the named layer state, the Layer States Manager closes.
- 6 Click Close to exit the Layer States Manager. Click OK to exit the Layer Properties Manager.

Layers toolbar



Command line: LAYER

Work with Colors

Color helps to group objects visually. You can assign colors to objects by layer or individually.

Set the Current Color

You can use color to help you identify objects visually. You can assign the color of an object either by layer or by specifying its color explicitly, independent of layer.

You can assign the color of an object either by layer or by specifying its color explicitly, independent of layer. Assigning colors by layer makes it easy to identify each layer within your drawing. Assigning colors explicitly provides additional distinctions between objects on the same layer. Color is also used as a way to indicate linewidth for color-dependent plotting.

You can use a variety of color palettes when assigning color to objects, including

- AutoCAD Color Index (ACI)
- True Color®, PANTONE®
- RAL™ Classic and RAL Design color books
- DIC® Color Guide
- Colors from imported color books.

ACI Colors

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

True Colors

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

Color Books

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

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

You install color books on your system by using the Files tab in the Options dialog box. Once a color book is loaded, you can select colors from the color book and apply them to objects in your drawings.

All objects are created using the current color, which is displayed in the Color control on the Properties toolbar. You can also set the current color with the Color control or the Select Color dialog box.

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

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

To set an ACI color for all new objects

- 1 On the Properties toolbar, click the Color control.
- 2 Click a color to draw all new objects in that color, or click Select Color to display the Select Color dialog box and do one of the following:
 - On the Index Color tab, click a color or enter the color name or number in the Color box.
 - On the Index Color tab, click BYLAYER to draw new objects in the color assigned to the current layer.
 - On the Index Color tab, click BYBLOCK to draw new objects in the current color until they are grouped into a block. When the block is inserted into the drawing, the objects in the block acquire the current color setting.
- 3 Click OK.

The Color control displays the current color.

Command line: COLOR

To set a true color for all new objects

- 1 On the Properties toolbar, click the Color control.
- 2 Click Select Color to display the Select Color dialog box.
- 3 In the Select Color dialog box, True Color tab, do one of the following:
 - Select the HSL color model in the Color Model box. Specify a color by entering a color value in the Color box or by specifying values in the Hue, Saturation, and Luminance boxes.
 - Select the RGB color model in the Color Model box. Specify a color by entering a color value in the Color box or by specifying values in the Red, Green, and Blue boxes.
- 4 Click OK.

The Color control displays the current color.

Command line: COLOR

To set a color from a color book for all new objects

- 1 On the Properties toolbar, click the Color control.
- 2 Click Select Color.
- 3 In the Select Color dialog box, Color Book tab, select a color book from the Color Book box.
- 4 Select a color by clicking on a color chip. To browse through the color book, use the up and down arrows on the color slider.
- 5 Click OK.

The Color control displays the current color.

Command line: COLOR

Change the Color of an Object

You can change the color of an object by reassigning it to another layer, by changing the color of the layer the object is on, or by specifying a color for the object explicitly.

You have three choices for changing the color of an object:

- Reassign the object to another layer with a different color. If an object's color is set to BYLAYER, and you reassign the object to a different layer, it acquires its color from the new layer.
- Change the color assigned to the layer that the object is on. If an object's color is set to BYLAYER, it acquires the color of its layer. When you change the color assigned to a layer, all objects on that layer assigned the BYLAYER color are updated automatically.
- Specify a color for an object to override the layer's color. You can specify the color of each object explicitly. If you want to override the layer-determined color of an object with a different one, change an existing object's color from BYLAYER to a specific color, such as red, or a true color.

If you want to set a specific color for all subsequently created objects, change the current color setting on the Properties toolbar from BYLAYER to a specific color.

To change the layer of an object

- 1 Select the objects whose layer you want to change.

- 2** On the Layers toolbar, click the Layer control.
- 3** Select the layer that you want to assign to the objects.



To change the color assigned to a layer

- 1** On the Layers toolbar, click the Layer Properties Manager button.
- 2** In the Layer Properties Manager, click the color you want to change.
- 3** In the Select Color dialog box, do one of the following:
 - On the Index tab, click a color or enter the ACI color number (1-255) or name in the Color box. Click OK.
 - On the True Color tab, select the HSL color model in the Color Model option and specify a color by entering a color value in the Color box or by specifying values in the Hue, Saturation, and Luminance boxes. Click OK.
 - On the Color Books tab, select a color book from the Color Book box; select a color by navigating the color book (using the up and down arrows) and clicking on a color chip. Click OK.
- 4** Click OK to close each dialog box.



Command line: LAYER

To change the color of objects, overriding the layer's color

- 1** Select the objects whose color you want to change.
- 2** Right-click one of the objects. Click Properties.
- 3** In the Properties palette, select Color.
An arrow is displayed in the right column.

- 4 Click the arrow and select a color from the list or click Select Color to display the Select Color dialog box. If you click Select Color, do one of the following:
 - On the Index tab, click a color or enter the ACI color number (1-255) or name in the Color box.
 - On the True Color tab, select the HSL color model in the Color Model option and specify a color by entering a color value in the Color box or by specifying values in the Hue, Saturation, and Luminance boxes. Click OK.
 - On the True Color tab, select the RGB color model in the Color Model box. Specify a color by entering a color value in the Color box or by specifying values in the Red, Green, and Blue boxes.
 - On the Color Books tab, select a color book from the Color Book box; select a color by navigating the color book (using the up and down arrows) and clicking on a color chip. Click OK.

Command line: PROPERTIES

Use Color Books

When assigning colors to objects, you can choose colors from color books that are loaded on your system. You can choose from a wide range of custom colors when using color books. Color books include third-party or user-defined files that contain named color swatches. These colors can be used to enhance presentation drawings as well as to optimize the variety of color used in your drawings. You can apply color book colors to objects in your drawings by using the Color Books tab in the Select Color dialog box.

Install Color Books

Color book files must contain an .acb file extension in order to be recognized by this program. To access color book colors from the Select Color dialog box, you must first copy your color book files to a specified color book location. On the Files tab of the Options dialog box, you can define the path where color book files are stored. Multiple locations can be defined for the color book path. These locations are saved in your user profile.

After loading a color book on your system, to access the new colors, you need to close the Select Color dialog box and then open it again. The new color book is displayed in the Color Book drop-down list on the Color Books tab. Once you have loaded a color book, you can apply any colors that are defined in the book to objects in your drawing.

Browse Color Books

Color books are organized alphabetically into pages that you can browse through. A page holds up to ten colors. If the color book you are browsing through is not organized into pages, the colors are arranged into pages, with each page containing up to seven colors.

To install a color book

- 1 Click Tools menu ► Options.
- 2 In the Options dialog box, Files tab, click Color Book Locations.
- 3 Click Add to add a color book location.
- 4 Enter the new location in the blank path box.
- 5 Click OK.

Command line: OPTIONS

To search for color swatches within a color book

- 1 Click Format menu ► Color.
- 2 In the Color dialog box, Color Books tab, select a color book from the Color Book drop-down list.
You must first select any color swatch in the color book in order to activate the Color edit box.
- 3 Under Color, enter the number of the color swatch you would like to locate and press TAB.
The Color edit box and the New color chip display the requested color or the color that is the closest match.
- 4 Click OK to apply the color.

Command line: COLOR

To change the default location of color book files

- 1 Click Tools menu ► Options.
- 2 In the Options dialog box, Files tab, double-click Color Book Locations.
- 3 Select the following default color book location:
C:\Program Files\{current AutoCAD release number}\support\color
- 4 Click the location name to edit the path.

- 5 Enter the new location in the path box.
- 6 Click OK.

Command line: OPTIONS

To define multiple folders for the color book path

- 1 Click Tools menu ► Options.
- 2 In the Options dialog box, Files tab, click Color Book Locations.
- 3 Click Add to add a color book location.
- 4 Enter the new location in the blank path box.
- 5 Click OK.

Command line: OPTIONS

Work with Linetypes

You can use linetypes to distinguish objects from one another visually and make your drawing easier to read.

Overview of Linetypes

A linetype is a repeating pattern of dashes, dots, and blank spaces displayed in a line or a curve. You assign linetypes to objects either by layer or by specifying the linetype explicitly, independent of layers.

In addition to choosing a linetype, you can set its scale to control the size of the dashes and spaces, and you can create your own custom linetypes.

NOTE These linetypes should not be confused with the hardware linetypes provided by some plotters. The two types of dashed lines produce similar results. Do not use both types at the same time, however, because the results can be unpredictable.

See also:

*"Custom Linetypes" in the *Customization Guide**

Load Linetypes

At the start of a project, you load the linetypes that are required for the project so that they are available when you need them.

If you want to know what linetypes are already available, you can display a list of linetypes that are loaded in the drawing or stored in an LIN (linetype definition) file.

This program includes the linetype definition files *acad.lin* and *acadiso.lin*. Which linetype file is appropriate depends on whether you use imperial or metric measurements.

- For imperial units, use the *acad.lin* file.
- For metric measurements, use the *acadiso.lin* file.

Both linetype definition files contain several complex linetypes.

If you select a linetype whose name begins with ACAD_ISO, you can use the ISO pen-width option when you plot.

You can remove unreferenced linetype information with PURGE or by deleting the linetype from the Linetype Manager. BYBLOCK, BYLAYER, and CONTINUOUS linetypes cannot be removed.

To load a linetype

- 1 Click Format menu ▶ Linetype.
- 2 In the Linetype Manager, click Load.
- 3 In the Load or Reload Linetypes dialog box, select a linetype. Click OK.
If the linetype you need is not listed, click File. In the Select Linetype File dialog box, select an LIN file whose linetypes you want to list and click. The dialog box displays the linetype definitions stored in the selected LIN file. Select a linetype. Click OK.
You can hold down CTRL to select several linetypes or SHIFT to select a range of linetypes.
- 4 Click OK.

Command line: LINETYPE

To list the linetypes loaded in the current drawing

- 1 On the Properties toolbar, click the Linetype control.

All loaded linetypes are displayed in the list. If necessary, use the scroll bar to see the whole list.

- 2 Click anywhere outside the box to close it.

To list the linetypes in a linetype definition file

- 1 Click Format menu ► Linetype.
- 2 In the Linetype Manager, click Load.
- 3 In the Load or Reload Linetypes dialog box, click File.
- 4 In the Select Linetype File dialog box, select an LIN (linetype definition) file whose linetypes you want to list. Click Open.
The dialog box displays the linetype definitions stored in the selected LIN file.
- 5 In the Load or Reload Linetypes dialog box, click Cancel.
- 6 Click Cancel to close the Linetype Manager.

Command line: LINETYPE

To unload an unused linetype

- 1 Click Format menu ► Linetype.
- 2 In the Linetype Manager, select a linetype. Click Delete.
The selected linetype is unloaded. Certain linetypes cannot be unloaded: BYLAYER, BYBLOCK, CONTINUOUS, and any linetypes currently in use.

Command line: LINETYPE

To purge an unused linetype

- 1 Click File menu ► Drawing Utilities ► Purge.
The Purge dialog box displays a tree view of object types with items that can be purged.
- 2 To purge unreferenced linetypes, use one of the following methods:
 - To purge all unreferenced linetypes, select Linetypes.
 - To purge specific linetypes, double-click Linetypes to expand the tree view. Then select the linetypes to be purged.

If the item you want to purge is not listed, select View Items You Cannot Purge.

- 3** You are prompted to confirm each item in the list. If you do not want to confirm each purge, clear the Confirm Each Item to Be Purged option.
- 4** Click Purge.
To confirm the purging of each item, respond to the prompt by choosing Yes or No, or Yes to All if more than one item is selected.
- 5** Click Close.

Command line: PURGE

Set the Current Linetype

All objects are created using the current linetype, which is displayed in the Linetype control on the Properties toolbar.

You can also set the current linetype with the Linetype control.

If the current linetype is set to BYLAYER, objects are created with the linetype assigned to the current layer.

If the current linetype is set to BYBLOCK, objects are created using the CONTINUOUS linetype until the objects are grouped into a block. When the block is inserted into the drawing, those objects acquire the current linetype setting.

If you do not want the current linetype to be the linetype assigned to the current layer, you can specify a different linetype explicitly.

The program does not display the linetype of certain objects: text, points, viewports, hatches, and blocks.

To set the linetype for all new objects

- 1** Click Format menu ► Linetype.
- 2** If you need to load additional linetypes, click Load, select one or more linetypes. Click OK.
You can hold down CTRL to select several linetypes or SHIFT to select a range of linetypes.
- 3** In the Linetype Manager, do one of the following:
 - Select a linetype and select Current to draw all new objects with that linetype.
 - Select BYLAYER to draw new objects in the linetype assigned to the current layer.

- Select BYBLOCK to draw new objects in the current linetype until they are grouped into a block. When the block is inserted into a drawing, the objects in the block acquire the current linetype setting.

4 Click OK.

NOTE The Linetype control displays the current linetype. If the linetype you want to use is already loaded, you can click the Linetype control and click the linetype to make it current.

Command line: LINETYPE

To change the linetype assigned to a layer

- 1 On the Layers toolbar, click the Layer Properties Manager button.
- 2 In the Layer Properties Manager, select the linetype name you want to change.
- 3 In the Select Linetype dialog box, select the linetype you want. Click OK.
- 4 Click OK again.

Layers toolbar



Command line: LAYER

Change the Linetype of an Object

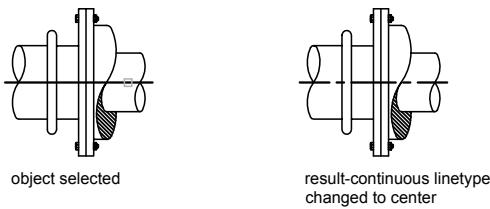
You can change the linetype of an object by reassigning it to another layer, by changing the linetype of the layer the object is on, or by specifying a linetype for the object explicitly.

You have three choices for changing the linetype of an object:

- Reassign the object to another layer with a different linetype. If an object's linetype is set to BYLAYER, and you reassign the object to a different layer, it acquires its linetype from the new layer.
- Change the linetype assigned to the layer that the object is on. If an object's linetype is set to BYLAYER, it acquires the linetype of its layer. When you change the linetype assigned to a layer, all objects on that layer assigned the BYLAYER linetype are updated automatically.

- Specify a linetype for an object to override the layer's linetype. You can specify the linetype of each object explicitly. If you want to override the layer-determined linetype of an object with a different one, change an existing object's linetype from BYLAYER to a specific linetype, such as DASHED.

If you want to set a specific linetype for all subsequently created objects, change the current linetype setting on the Properties toolbar from BYLAYER to a specific linetype.



To change the default linetype assigned to a layer

- 1 On the Layers toolbar, click the Layer Properties Manager button.
- 2 In the Layer Properties Manager, select the linetype name you want to change.
- 3 In the Select Linetype dialog box, select a linetype.
If the linetype you need is not listed, click Load. Select a linetype and click OK, or click File to open additional linetype definition (LIN) files.
- 4 Click OK to exit each dialog box.

Layers toolbar



Command line: LAYER

To change the linetype of an object, overriding the layer's linetype

- 1 Select the objects whose linetype you want to change.
- 2 On the Properties toolbar, click the Linetype control.
- 3 Choose the linetype that you want to assign to the objects.

Command line: LINETYPE

Control Linetype Scale

You can use the same linetype at different scales by changing the linetype scale factor either globally or individually for each object.

By default, both global and individual linetype scales are set to 1.0. The smaller the scale, the more repetitions of the pattern are generated per drawing unit. For example, with a setting of 0.5, two repetitions of the pattern in the linetype definition are displayed for each drawing unit. Short line segments that cannot display one full linetype pattern are displayed as continuous. You can use a smaller linetype scale for lines that are too short to display even one dash sequence.

The Linetype Manager displays the Global Scale Factor and Current Object Scale.

- The Global Scale Factor value controls the LTSCALE system variable, which changes the linetype scale globally for both new and existing objects.
- The Current Object Scale value controls the CELTSCALE system variable, which sets the linetype scale for new objects.

The CELTSCALE value is multiplied by the LTSCALE value to get the displayed linetype scale. You can easily change linetype scales in your drawing either individually or globally.

In a layout, you can adjust the scaling of linetypes in different viewports with PSLTSCALE.

To change the linetype scale of selected objects

- 1 Select the objects whose linetype scale you want to change.
- 2 Right-click one of the objects. Click Properties.
- 3 In the Properties palette, select Linetype Scale and enter the new value.

Standard toolbar



Command line: PROPERTIES

To set the linetype scale for new objects

- 1 Click Format menu ► Linetype.
- 2 In the Linetype Manager, click Show Details to expand the dialog box.
- 3 Enter a new value for Current Object Scale.
- 4 Click OK.

Command line: LINETYPE

To change linetype scale globally

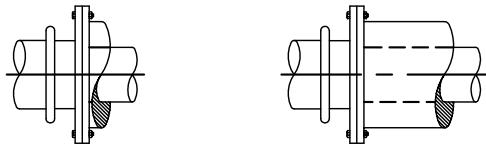
- 1 Click Format menu ► Linetype.
- 2 In the Linetype Manager, click Show Details to expand the dialog box.
- 3 Enter a new value for Global Scale Factor.
- 4 Click OK.

Command line: LINETYPE

Display Linetypes on Short Segments and Polylines

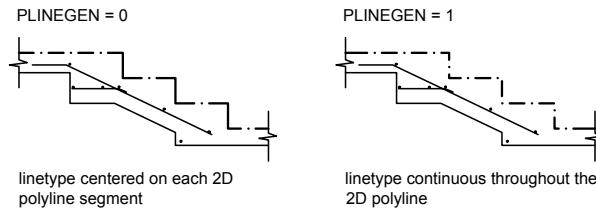
You can center the pattern of a linetype on each segment of a polyline, and you can control how the linetype is displayed on short segments.

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 315.

For polylines, you can specify whether a linetype pattern is centered on each segment or is continuous across vertices throughout the entire length of the polyline. You do this by setting the PLINEGEN system variable.



To set the linetype display for all new polylines

- 1 At the Command prompt, enter **plinegen**.
- 2 Enter **1** to make the linetype pattern continue throughout the entire length of two-dimensional polylines, or enter **0** to center the linetype pattern on each segment.

Command line: PLINEGEN

To change the linetype display of existing polylines

- 1 Double-click the polyline whose linetype display you want to change.
- 2 In the Properties palette, click Linetype Generation and select Enabled or Disabled.

Standard toolbar



Command line: PROPERTIES

Control Lineweights

You can further differentiate objects in your drawing by controlling their lineweights both in the drawing display and in plotting.

Overview of Lineweights

Using lineweights, you can create heavy and thin lines to show cuts in sections, depth in elevations, dimension lines and tick marks, and differences in details. For example, by assigning varying lineweights to different layers, you can easily differentiate between new, existing, and demolition construction.

Lineweights are not displayed unless the LWT button on the status bar is selected.

All objects except TrueType fonts, raster images, points, and solid fills (2D solids) can display linewidth. In a plan view, wide polylines override any width value applied by a linewidth. Wide polylines display lineweights only when the polyline is being viewed in a view other than Plan. You can export drawings to other applications or cut objects to the Clipboard and retain linewidth 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 linewidth; instead, use a polyline with a width of 0.5 inches to represent the object.

You can also plot objects in your drawing with custom linewidth values. Use the Plot Style Table Editor to adjust the fixed linewidth values to plot at a new value.

Lineweight Scale in Drawings

Objects with a linewidth are plotted with the exact width of the assigned linewidth 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 linewidth value of 0.025 mm or less is displayed as one pixel in model space and is plotted at the thinnest linewidth available on the specified plotting device. Linewidth values that you enter on the command line are rounded to the nearest predefined value.

You set the linewidth units and the default value in the Lineweight Settings dialog box. You can access the Lineweight Settings dialog box by using the LWEIGHT command, by right-clicking the LWT button on the status bar and choosing Settings, or by choosing Lineweight Settings on the Display tab in the Options dialog box.

See also:

["Draw Polylines" on page 419](#)

To assign width to polylines

- 1 Click Draw menu ► Polyline.
- 2 Specify the start point for the first segment in the polyline.
- 3 Enter **w** (width).

- 4 Enter a value for the width at the start of the line segment.
- 5 Enter a value for the width at the end of the line segment.
- 6 Specify the endpoint of the first segment of the polyline.
- 7 Enter **w** to specify different widths for the next segment, or press **ENTER** to end the command.

Draw toolbar



Command line: PLINE

To assign a linewidth to a layer

- 1 Click the Format menu ► Layer.
- 2 In the Layer Properties Manager, select a layer. Click the linewidth associated with that layer.
- 3 In the Lineweight dialog box, select a linewidth from the list.
- 4 Click OK to close each dialog box.

Command line: LAYER

To set the display scale of lineweights on the Model tab

- 1 Click Format menu ► Lineweight.
- 2 In the Lineweight Settings dialog box, under Adjust Display Scale, move the slider to change the scale.
- 3 Click OK.

Command line: LAYER, LWEIGHT

Display Lineweights

Lineweights are displayed differently in model space than in a paper space layout.

- In model space, a 0-value linewidth is displayed as one pixel, and other lineweights use a pixel width proportional to their real-unit value.

- In a paper space layout, lineweights are displayed in the exact plotting width.

Display Lineweights in Model Space

In model space, weighted lines that are joined form a beveled joint with no end caps. You can use plot styles to apply different joins and endcap styles to objects with lineweights.

NOTE Different styles of endcaps and joins of objects with linewidth are displayed only in a full plot preview.

Lineweight display in model space does not change with the zoom factor. For example, a linewidth 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 linewidths on objects to appear thicker or thinner on the Model tab, use LWEIGHT to set their display scale. Changing the display scale does not affect the linewidth plotting value.

Regeneration time increases with linewidths that are represented by more than one pixel. Turn off the display of linewidths to optimize performance of the program. On the Model tab, you can turn the display of linewidths on or off by clicking LWT on the status bar.

Display Lineweights in Layouts

In layouts and plot preview, linewidths are displayed in real-world units, and linewidth display changes with the zoom factor. You can control linewidth plotting and scaling in your drawing in the Plot dialog box, Plot Settings tab.

In layouts, you can turn the display of linewidths on or off by clicking LWT on the status bar. This setting does not affect the plotting of linewidths.

To display linewidths

Use one of the following methods:

- Toggle LWT on the status bar.
- Select or clear Display Lineweight in the Lineweight Settings dialog box.
- Set the LWDISPLAY system variable to 0 or 1 to turn linewidth display off or on.

Command line: LWEIGHT

Set the Current Lineweight

The current linewidth is the linewidth used for any objects you draw until you make another linewidth current.

All objects are created using the current linewidth, which is displayed in the Lineweight control on the Properties toolbar. You can also set the current linewidth with the Lineweight control.

If the current linewidth is set to BYLAYER, objects are created with the linewidth assigned to the current layer.

If the current linewidth is set to BYBLOCK, objects are created using the default linewidth setting until the objects are grouped into a block. When the block is inserted into the drawing, it acquires the current linewidth setting.

If you do not want the current linewidth to be the linewidth assigned to the current layer, you can specify a different linewidth explicitly.

Objects in drawings created in an earlier release of AutoCAD are assigned the linewidth value of BYLAYER, and all layers are set to DEFAULT. Lineweight assigned to objects is displayed as a solid fill drawn in the object's assigned color.

To make a linewidth current for creating objects

- 1 Click Format menu ► Lineweight.
- 2 In the Lineweight Settings dialog box, select a linewidth.
- 3 Click OK.

NOTE To display the linewidth at its current setting, the Display Lineweight option must be selected in the Display Lineweight dialog box.

Command line: LAYER, LWEIGHT

Change the Lineweight of an Object

You can change the linewidth of an object by reassigning it to another layer, by changing the linewidth of the layer the object is on, or by specifying a linewidth for the object explicitly.

You have three choices for changing the linewidth of an object:

- Reassign the object to another layer with a different linewidth. If an object's linewidth is set to BYLAYER, and you reassign the object to a different layer, it acquires its linewidth from the new layer.
- Change the linewidth assigned to the layer that the object is on. If an object's linewidth is set to BYLAYER, it acquires the linewidth of its layer. When you change the linewidth assigned to a layer, all objects on that layer assigned the BYLAYER linewidth are updated automatically.
- Specify a linewidth for an object to override the layer's linewidth. You can specify the linewidth of each object explicitly. If you want to override the layer-determined linewidth of an object with a different one, change an existing object's linewidth from BYLAYER to a specific linewidth.

If you want to set a specific linewidth for all subsequently created objects, change the current linewidth setting on the Properties toolbar from BYLAYER to a specific linewidth.

To change the line width of a polyline, donut, rectangle, or polygon

- 1 Click Modify menu ▶ Object ▶ Polyline.
- 2 Select one or more polyline objects.
- 3 Enter **w** (Width) and enter a new width for all segments.
- 4 Press ENTER to end the command.

Modify II toolbar



Command line: PEDIT

Control the Display Properties of Certain Objects

You can control how overlapping objects and certain other objects are displayed and plotted.

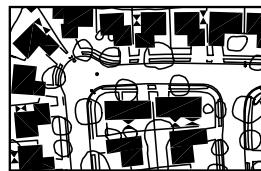
Control the Display of Polyline, Hatches, Gradient Fills, Lineweights, and Text

You can simplify the display of certain kinds of objects in order to speed performance.

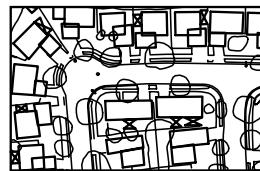
Display performance is improved when wide polylines and donuts, solid-filled polygons (two-dimensional solids), hatches, gradient fills, and text are displayed in simplified form. Simplified display also increases the speed of creating test plots.

Turn Off Solid Fill

When you turn off Fill mode, wide polylines, solid-filled polygons, gradient fill, and hatches are displayed in outline form. Except for patterned hatches and gradient fills, solid fill is automatically turned off for hidden view and nonplan views in three dimensions.



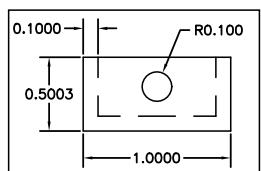
Fill mode on



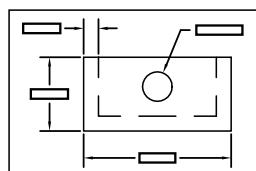
Fill mode off

Use Quick Text

When you turn on Quick Text mode in drawings that contain a lot of text using complex fonts, only a rectangular frame defining the text is displayed or plotted.



Quick Text mode off



Quick Text mode on

Turn Off Lineweights

Any linewidth width that is represented by more than one pixel may slow down performance. If you want to improve display performance, turn lineweights off. You can turn lineweights on and off by choosing the LWT button on the status bar or by using the Lineweight Settings dialog box. Lineweights are always plotted at their real-world value whether their display is turned on or off.

Update the Display

New objects automatically use the current settings for displays of solid fill and text. Except for lineweights, to update the display of existing objects using these settings, you must use REGEN.

See also:

- “Use Layers to Manage Complexity” on page 278
- “Display Lineweights” on page 319
- “Use TrueType Fonts” on page 833

To turn the display of solid fill on or off

- 1 Click Tools menu ► Options.
- 2 In the Options dialog box, Display tab, under Display Performance, select Apply Solid Fill.
A check mark indicates that Fill mode is on.
- 3 Click OK.
- 4 To display your changes, click View menu ► Regen.

Command line: OPTIONS, REGEN

To turn the display of text on or off

- 1 Click Tools ► Options.
- 2 In the Options dialog box, Display tab, under Display Performance, select Show Text Boundary Frame Only.
The check mark indicates that text is displayed as a rectangular frame.
- 3 Click OK.
- 4 To display your changes, click View menu ► Regen.

Command line: OPTIONS, REGEN

To turn lineweights on or off

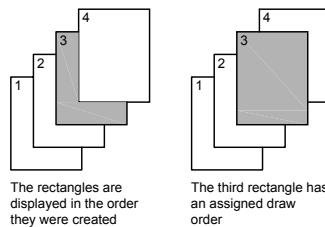
- 1 Click Format menu ► Lineweight.
- 2 In the Lineweight Settings dialog box, select or clear Display Lineweight.
- 3 Click OK.

Command line: LWEIGHT

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.



NOTE Overlapping objects cannot be controlled between model space and paper space. They can be controlled only within the same space.

To change the draw order of overlapping objects

- 1 On the Draw Order toolbar, click one of the draw order options.
- 2 Select the object(s) whose draw order you want to modify and press ENTER.
- 3 Select the reference object(s) and press ENTER. (This step is necessary only for the Bring Above Objects and Send Under Objects options.)

Tools menu Draw Order

Command line: DRAWORDER

Shortcut menu: Select an object and right-click. Click Draw Order

Use Precision Tools

I 8

You can use a variety of precision drawing tools to help you produce accurate drawings quickly and without performing tedious calculations.

In this chapter

- Use Coordinates and Coordinate Systems (UCS)
- Use Dynamic Input
- Snap to Locations on Objects (Object Snaps)
- Restrict Cursor Movement
- Combine or Offset Points and Coordinates
- Specify Distances
- Extract Geometric Information from Objects
- Use a Calculator

Use Coordinates and Coordinate Systems (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 on the command line. When Dynamic Input is on, you can enter coordinate values in tooltips near the cursor. You can enter two-dimensional coordinates as either Cartesian (X,Y) or polar coordinates.

Cartesian and Polar Coordinates

A Cartesian coordinate system has three axes, X , Y , and Z . When you enter coordinate values, you indicate a point's distance (in units) and its direction (+ or -) along the X , Y , and Z axes relative to the coordinate system origin (0,0,0).

In 2D, you specify points on the XY plane, also called the *workplane*. The workplane is similar to a flat sheet of grid paper. The X value of a Cartesian coordinate specifies horizontal distance, and the Y value specifies vertical distance. The origin point (0,0) indicates where the two axes intersect.

Polar coordinates use a distance and an angle to locate a point. With both Cartesian and polar coordinates, you can enter absolute coordinates based on the origin (0,0), or relative coordinates based on the last point specified.

Another method of entering a relative coordinate is by moving the cursor to specify a direction and then entering a distance directly. This method is called direct distance entry.

You can enter coordinates in scientific, decimal, engineering, architectural, or fractional notation. You can enter angles in grads, radians, surveyor's units, or degrees, minutes, and seconds. The UNITS command controls unit format.

Display Coordinates on the Status Bar

The current cursor location is displayed as a coordinate value on the status bar.

411,162,0

There are three types of coordinate display: static, dynamic, and distance and angle.

- **Static display.** Updates only when you specify a point.
- **Dynamic display.** Updates as you move the cursor.
- **Distance and angle display.** Updates the relative distance (*distance<angle*) as you move the cursor. This option is available only when you draw lines or other objects that prompt for more than one point.

See also:

- [Enter Cartesian Coordinates](#)
- [Enter Polar Coordinates](#)
- [Enter 3D Coordinates](#)
- [Use Dynamic Input](#)

To display the coordinate values of a point

1 Click Tools menu ▶ Inquiry ▶ ID Point.

2 Select the location you want to identify.

The *X,Y,Z* coordinate values are displayed on the command line.

Inquiry toolbar



Command line: ID

To visually locate a point

1 Click Tools menu ▶ Inquiry ▶ ID Point.

2 On the command line, enter the coordinate values of the point you want to locate.

If the BLIPMODE system variable is on, a blip (a small cross) is displayed at the point location.

Inquiry toolbar



Command line: ID

To change the coordinate display on the status bar

Use one of the following methods:

- Click the coordinate display at the Specify Next Point prompt.
- Press CTRL+L.
- Set the COORDS system variable to 0 for static display, 1 for dynamic display, or 2 for distance and angle display.

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.

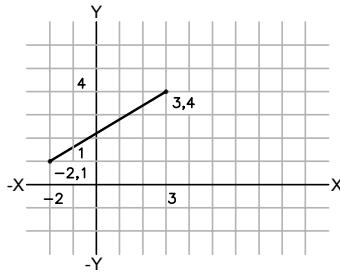
The following example draws a line beginning at an *X* value of -2, a *Y* value of 1, and an endpoint at 3,4. Enter the following in the tooltip:

Command: **line**

From point: **#-2,1**

To point: **#3,4**

The line is located as follows:



Relative coordinates are based on the last point entered. Use relative coordinates when you know the location of a point in relation to the previous point.

To specify relative coordinates, precede the coordinate values with an @ sign. For example, entering **@3,4** specifies a point 3 units along the X axis and 4 units along the Y axis from the last point specified.

The following example draws the sides of a triangle. The first side is a line starting at the absolute coordinates $-2,1$ and ending at a point 5 units in the X direction and 0 units in the Y direction. The second side is a line starting at the endpoint of the first line and ending at a point 0 units in the X direction and 3 units in the Y direction. The final line segment uses relative coordinates to return to the starting point.

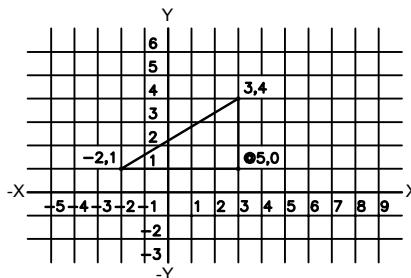
Command: **line**

From point: **#-2,1**

To point: **@5,0**

To point: **@0,3**

To point: **@-5,-3**



To enter absolute Cartesian coordinates (2D)

At a prompt for a point, enter coordinates in the tooltip using the following format:

x, y

If Dynamic Input is turned off, enter coordinates on the command line using the following format:

x, y

To enter relative Cartesian coordinates (2D)

At a prompt for a point, enter coordinates using the following format:

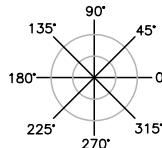
@ x, y

Enter Polar Coordinates

You can use absolute or relative polar coordinates (distance and angle) to locate points when creating objects.

To use polar coordinates to specify a point, enter a distance and an angle separated by an angle bracket (<).

By default, angles increase in the counterclockwise direction and decrease in the clockwise direction. To specify a clockwise direction, enter a negative value for the angle. For example, entering **1<315** locates the same point as entering **1<-45**. You can change the angle conventions for the current drawing with 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.

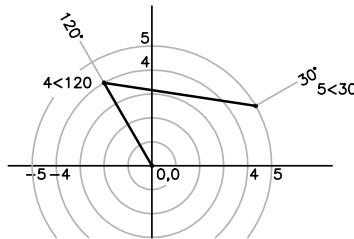
The following example shows two lines drawn with absolute polar coordinates using the default angle direction setting. Enter the following in the tooltip:

Command: **line**

From point: #**0,0**

To point: #**4<120**

To point: #**5<30**



Relative coordinates are based on the last point entered. Use relative coordinates when you know the location of a point in relation to the previous point.

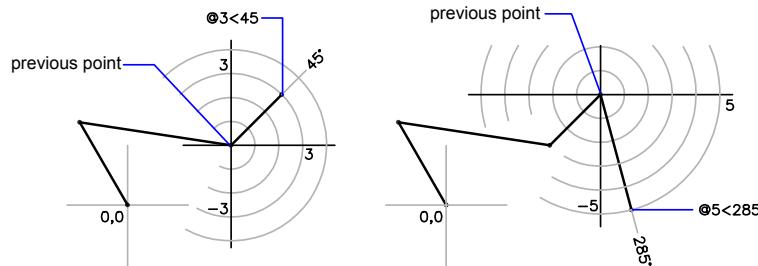
To specify relative coordinates, precede the coordinate values with an @ sign. For example, entering @1<45 specifies a point at a distance of 1 unit from the last point specified at an angle of 45 degrees from the X axis.

The following example shows two lines drawn with relative polar coordinates. In each illustration, the line begins at the location labeled as the previous point.

Command: **line**

From point: @**3<45**

To point: @**5<285**



To enter absolute polar coordinates (2D)

- At a prompt for a point, enter coordinates in the tooltip using the following format:

distance < angle

If Dynamic Input is turned off, enter coordinates on the command line using the following format:

distance < angle

To enter relative polar coordinates (2D)

- At a prompt for a point, enter coordinates using the following format:

@ distance < angle

Enter 3D Coordinates

Cartesian, cylindrical, or spherical coordinates locate points when you are creating objects in 3D.

Enter 3D Cartesian Coordinates

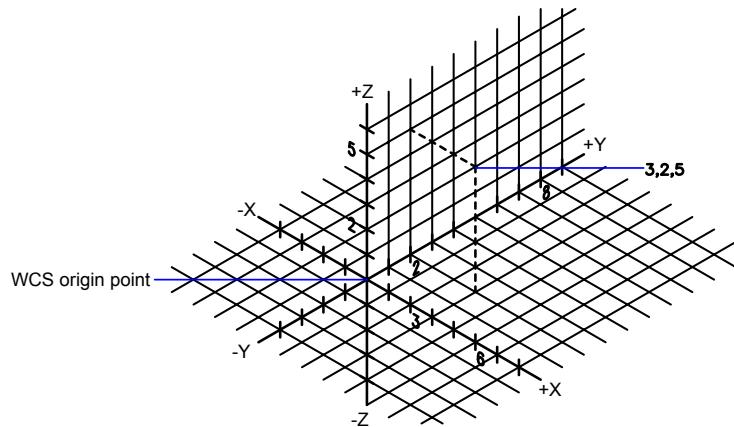
3D Cartesian coordinates specify a precise location by using three coordinate values: *X*, *Y*, and *Z*.

Entering 3D Cartesian coordinate values (*X,Y,Z*) is similar to entering 2D coordinate values (*X,Y*). In addition to specifying *X* and *Y* values, you also specify a *Z* value using the following format:

X,Y,Z

NOTE For the following examples, it is assumed that dynamic input is turned off or that the coordinates are entered on the command line. With Dynamic Input, you specify absolute coordinates with the # prefix.

In the illustration below, the coordinate values of 3,2,5 indicates a point 3 units along the positive *X* axis, 2 units along the positive *Y* axis, and 5 units along the positive *Z* axis.



Use Default Z Values

When you enter coordinates in the format X,Y , the Z value is copied from the last point you entered. As a result, you can enter one location in the X,Y,Z format and then enter subsequent locations using the X,Y format with the Z value remaining constant. For example, if you enter the following coordinates for a line

From point: **0,0,5**
To point: **3,4**

both endpoints of the line will have a Z value of 5. When you begin or open any drawing, the initial default value of Z is 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 on the command line, no prefix is necessary.

Digitize Coordinates

When you enter coordinates by digitizing, the UCS Z value for all coordinates is 0. You can use ELEV to set a default height above or below the $Z = 0$ plane for digitizing without moving the UCS.

To enter absolute coordinates (3D)

At a prompt for a point, enter coordinates in the tooltip using the following format:

x,y,z

If Dynamic Input is turned off, enter coordinates on the command line using the following format:

x , y , z

To enter relative coordinates (3D)

At a prompt for a point, enter coordinates using the following format:

@*x,y,z*

Enter Cylindrical Coordinates

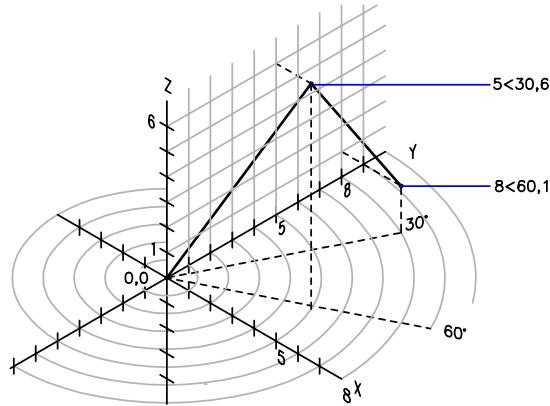
3D cylindrical coordinates describe a precise location by a distance from the UCS origin in the XY plane, an angle from the X axis in the XY plane, and a Z value.

Cylindrical coordinate entry is the 3D equivalent of 2D polar coordinate entry. It specifies an additional coordinate on an axis that is perpendicular to the XY plane. Cylindrical coordinates define points by a distance in the XY plane from the UCS origin, an angle from the X axis in the XY plane, and a Z value. You specify a point using the following syntax:

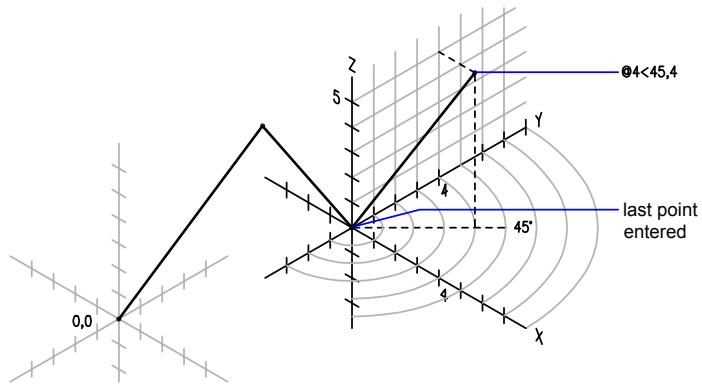
X <[angle from X axis], Z

NOTE For the following examples, it is assumed that dynamic input is turned off or that the coordinates are entered on the command line. With Dynamic Input, you specify absolute coordinates with the # prefix.

In the illustration below, 5<60,6 indicates a point 5 units from the origin of the current UCS, 60 degrees from the X axis in the XY plane, and 6 units along the Z axis. 8<30,1 indicates a point 8 units from the origin of the current UCS in the XY plane, 30 degrees from the X axis in the XY plane, and 1 unit along the Z axis.



When you need to define a point based on a previous point, you can enter relative cylindrical coordinate values with the @ sign. In the illustration below, @ $4<45,5$ indicates a point 4 units in the XY plane from the last point entered, at an angle of 45 degrees from the positive X direction, and extending 5 units in the Z direction.



To enter relative cylindrical coordinates

At a prompt for a point, enter the coordinate values using the following format:

@ x < angle from the X axis , z

For example, @4<60,2 represents a location that is 4 units along the X axis from the last point measured at 60 degrees from the positive X axis and at 2 units in the positive Z direction.

Enter Spherical Coordinates

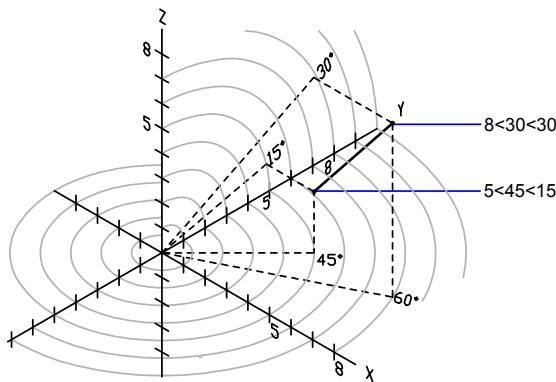
3D spherical coordinates specify a location by a distance from the origin of the current UCS, an angle from the X axis in the XY plane, and an angle from the XY plane.

Spherical coordinate entry in 3D is similar to polar coordinate entry in 2D. You locate a point by specifying its distance from the origin of the current UCS, its angle from the X axis (in the XY plane), and its angle from the XY plane, each angle preceded by an open angle bracket (<) as in the following format:

X <[angle from X axis]<[angle from XY plane]

NOTE For the following examples, it is assumed that dynamic input is turned off or that the coordinates are entered on the command line. With Dynamic Input, you specify absolute coordinates with the # prefix.

In the following illustration, 8<60<30 indicates a point 8 units from the origin of the current UCS in the XY plane, 60 degrees from the X axis in the XY plane, and 30 degrees up the Z axis from the XY plane. 5<45<15 indicates a point 5 units from the origin, 45 degrees from the X axis in the XY plane, and 15 degrees up from the XY plane.



When you need to define a point based on a previous point, enter the relative spherical coordinate values by preceding them with the @ sign.

To enter relative spherical coordinates

- At a prompt for a point, enter the coordinate values using the following format:

@x<angle from the x axis<angle from the xy plane

For example, @4<60<30 represents a location that is 4 units from the last point measured at 60 degrees from the positive X axis in the XY plane and at 30 degrees from the XY plane.

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.

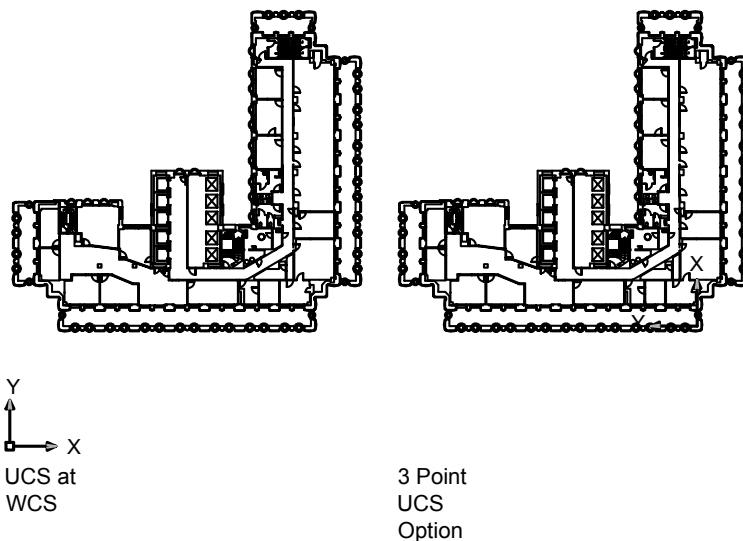
Normally in 2D views, the WCS X axis is horizontal and the Y axis is vertical. The WCS *origin* is where the X and Y axes intersect (0,0). All objects in a drawing file are defined by their WCS coordinates. However, it is usually more convenient to create and edit objects based on the movable UCS.

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 Tools menu > New UCS > Origin.

- 2 Specify a point for the new origin.

The UCS origin (0,0) is redefined at the point you specify.

UCS toolbar



Command line: UCS

To change the rotation angle of the UCS

- 1 Click Tools menu ► New UCS ► Z.

- 2 Specify a rotation angle.

UCS toolbar



Command line: UCS

To restore the UCS to be coincident with the WCS

- 1 Click Tools menu ► Named UCS.

- 2 In the UCS dialog box, Named UCSS tab, select World.

- 3 Click Set Current.

- 4 Click OK.

UCS toolbar



Command line: UCSMAN

To restore the previous UCS

- 1 Click Tools menu ► Named UCS.

- 2 In the UCS dialog box, Named UCSS tab, select Previous.

- 3 Click Set Current.

- 4 Click OK.

UCS toolbar



Command line: UCSMAN

To save a UCS

- 1 Click Tools menu ► Named UCS.

The new UCS is displayed in the UCS list as UNNAMED.

- 2 In the UCS dialog box, Named UCSS tab, select UNNAMED and enter a new name. (You can also select UNNAMED, and right-click. Click Rename.)
- 3 Click OK.

You can use up to 255 characters, including letters, digits, and the special characters dollar sign (\$), hyphen (-), and underscore (_). All UCS names are converted to uppercase.

UCS toolbar



Command line: UCSMAN

To restore a named UCS

- 1 Click Tools menu ► Named UCS.

- 2 In the UCS dialog box, Named UCSS tab, you can view the origin and axis direction of a listed UCS. Select the UCS name. Click Details.

When you have finished viewing the list, click OK to return to the UCS dialog box.

- 3 Select the coordinate system you want to restore. Click Set Current.
- 4 Click OK.

UCS toolbar



Command line: UCSMAN

To rename a UCS

- 1 Click Tools menu ► Named UCS.
- 2 In the UCS dialog box, Named UCSs tab, select the coordinate system you want to rename. (You can also select UNNAMED, and right-click. Click Rename.)
- 3 Enter a new name.
- 4 Click OK.

UCS toolbar



Command line: UCSMAN

To delete a named UCS

- 1 Click Tools menu ► Named UCS.
- 2 In the UCS dialog box, Named UCSs tab, select the UCS you want to delete.
- 3 Press DELETE.

You cannot delete the current UCS or a UCS with the default name UNNAMED.

UCS toolbar



Command line: UCSMAN

Specify Workplanes in 3D (UCS)

Control of the user coordinate system is essential for effective 3D modeling.

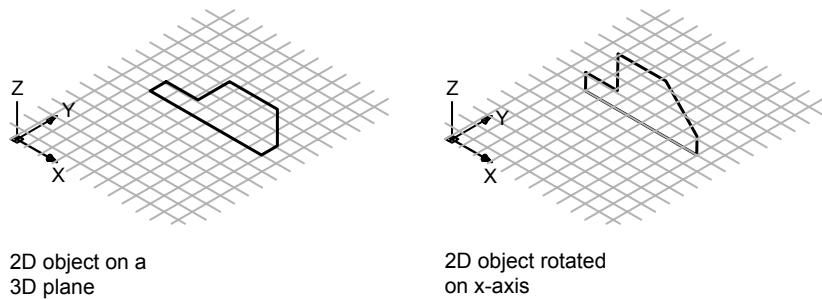
Understand the User Coordinate System in 3D

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:

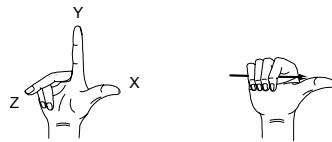
- Establish the workplane in on which to create and modify objects
- Establish the workplane that contains the grid display and grid snap
- Establish a new UCS Z axis about which to rotate objects in 3D
- Determine up and down directions as well as horizontal and vertical for Ortho mode, polar tracking, 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.



NOTE By default, when you specify a view in 3D, it is established relative to the fixed WCS rather than the movable UCS.

To enter coordinates relative to the WCS

- Precede coordinate values with an asterisk (*).

Entering **@*2,0,0** specifies a point two units in the *X* direction of the last point entered relative to the WCS. Entering **@2,0,0** specifies a point two units in the *X* direction of the last point entered relative to the UCS.

In practice, most coordinates are entered relative to the UCS rather than the WCS.

To specify a new UCS with three points

- 1 Click Tools menu ► New UCS ► 3 Point.



- 2 Specify a new origin point. This is the point (0,0,0) on the new UCS.
- 3 Specify a point on the positive *X* axis of the new UCS.
- 4 Specify a point on the positive *XY* plane of the new UCS.

To rotate the UCS around a major axis

- 1 Click Tools menu ► New UCS ► Z (or, alternatively, can click X or Y).

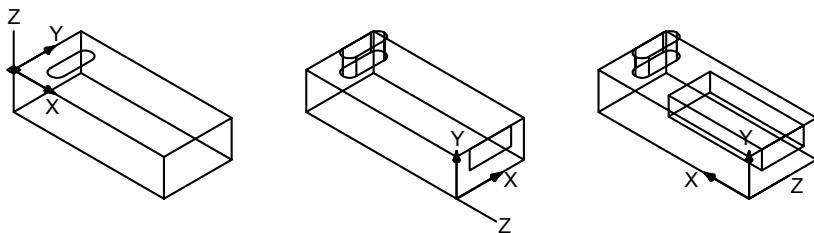


- 2 Enter a rotation angle. A common rotation angle is 90.

Control the User Coordinate System in 3D

Several methods are available for manipulating the user coordinate system in 3D. You can also save and restore user coordinate system orientations.

You define a user coordinate system (UCS) to change the location of the 0,0,0 origin point, the location and rotation of the XY plane, and the orientation of the XY plane or Z axis. You can locate and orient a UCS anywhere in 3D space, and you can define, save, and recall as many saved UCS locations as you require.



Moved UCS

Rotated UCS

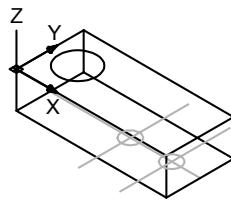
Roriated
Z-axis

If multiple viewports are active, you can assign a different UCS to each viewport. With the UCSVP system variable turned on, you can lock a UCS to a viewport, automatically restoring the UCS each time that viewport is made current.

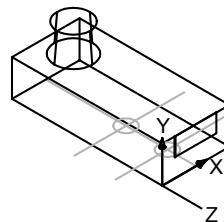
Define the UCS Location

You can define a UCS in the following ways:

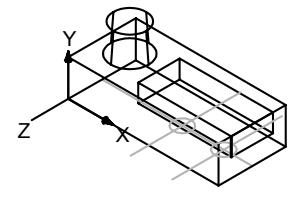
- Specify a new origin (one point), new X axis (two points), or new XY plane (three points).



UCS origin

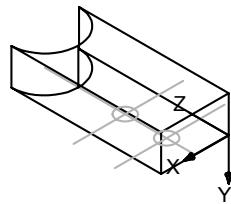


UCS x-axis option

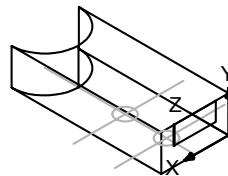


UCS 3-point option

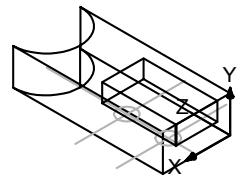
- Align the UCS by selecting a face on a 3D solid object. The selection can be on a face or on an edge of the solid.



UCS face
option

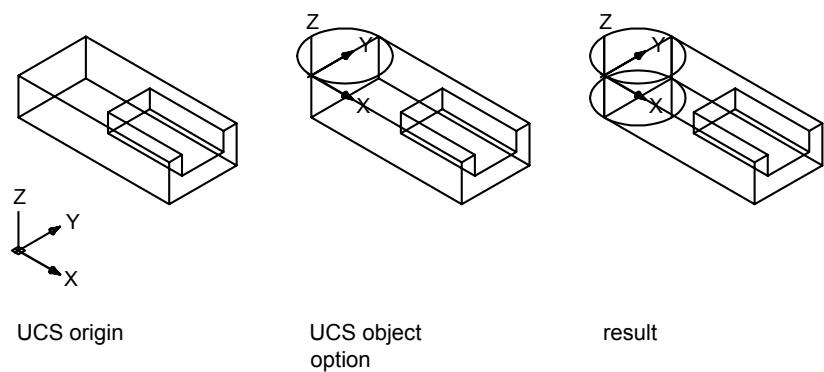


UCS x-flip option

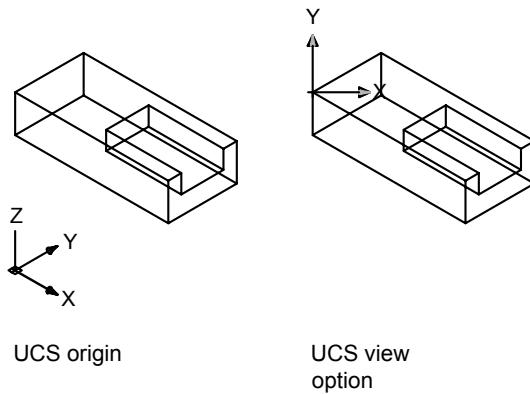


Result

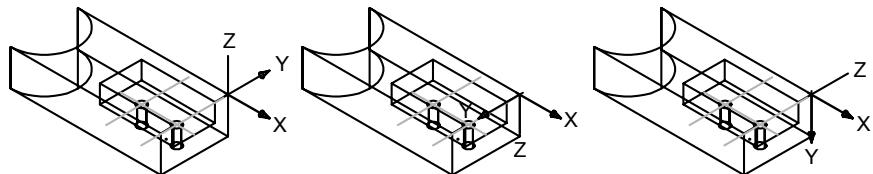
- Align the new UCS with an existing object. The origin of the UCS is located at the vertex nearest to where the object was selected.



- Align the new UCS with the current viewing direction.



- Rotate the current UCS around any of its three major axes.

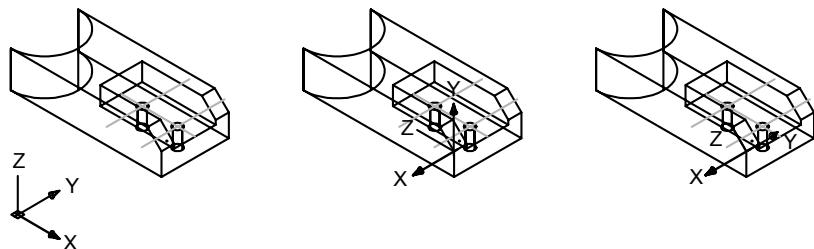


New UCS origin

UCS rotated
180 degrees on
the x-axis

UCS rotated 90
degrees on the
x-axis

- Reorient the XY plane by specifying a new Z axis.



UCS at
WCS

UCS ZAxis
option

UCS rotated
ZAxis option

Use UCS Presets

If you do not want to define your own UCS, you can choose from several preset coordinate systems. The images on the Orthographic UCSs tab of the (Named) UCS dialog box show the available choices.

Change the Default Elevation

The ELEV command sets the default Z value for new objects above or below the XY plane of the current UCS. This value is stored in the ELEVATION system variable.

NOTE Generally, it is recommended that you leave the elevation set to zero and control the XY plane of the current UCS with the UCS command.

Change the UCS in Paper Space

You can define a new UCS in paper space just as you can in model space; however, the UCS in paper space is restricted to 2D manipulation. Although you can enter 3D coordinates in paper space, you cannot use 3D viewing commands such as PLAN and VPOINT.

Save and Restore UCS Locations by Name

If you plan to work extensively in 3D, you can save named UCS locations, each having a different origin and orientation, for various construction requirements. You can relocate, save, and recall as many UCS orientations as you require.

To define a new UCS origin in 3D

- 1 Click Tools menu ► New UCS ► Origin.
- 2 Specify a point for the new origin.

The UCS origin (0,0,0) is redefined at the point you specify.

UCS toolbar



Command line: UCS

To define a new UCS with a particular Z axis

- 1 Click Tools menu ► New UCS ► Z Axis Vector.
- 2 Specify a point for the new origin.
The UCS origin (0,0,0) is redefined at the point you specify.
- 3 Specify a point that lies on the positive Z axis.

UCS toolbar



Command line: UCS

To define a new UCS with particular X and Y axes

- 1 Click Tools menu ► New UCS ► 3 Point.
- 2 Specify a point for the new origin.
The UCS origin (0,0,0) is redefined at the point you specify.
- 3 Specify a point that lies on the positive X axis.
- 4 Specify a point that lies on the positive Y axis.

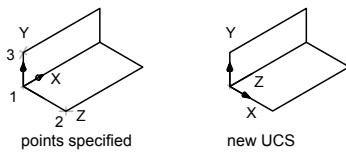
UCS toolbar



Command line: UCS

To shift the XY plane

- 1 Click Tools menu ► New UCS ► 3 Point.
 - 2 Specify an origin point for the new UCS (1).
For example, in a large drawing, you might specify an origin point near the area in which you want to work.
 - 3 Specify a point to indicate the horizontal orientation of the new UCS (2). This point should be on the positive portion of the new X axis.
 - 4 Specify a point to indicate the vertical orientation of the new UCS (3). This point should be on the positive portion of the new Y axis.
- The UCS, including grid, shifts to represent the X and Y axes you have specified.



UCS toolbar



Command line: UCS

To select a UCS preset

- 1 Click Tools menu ► Orthographic UCS ► Preset.
- 2 In the UCS dialog box, Orthographic UCSS tab, select a UCS orientation from the list.
- 3 Click Set Current.
- 4 Click OK.

The UCS changes to the selected option.

UCS II toolbar



Command line: UCSMAN

To restore the location and orientation of the previous UCS

- On the command line, enter **ucs**. Then enter **p** (Previous).

The previous UCS is restored.

UCS toolbar



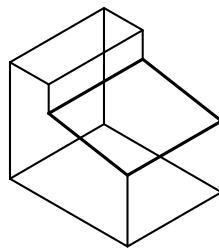
Command line: UCS

Use the Dynamic UCS with Solid Models

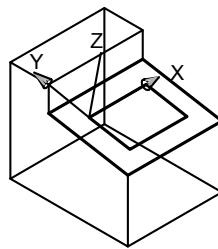
With the dynamic UCS feature, you can temporarily and automatically align the XY plane of the UCS with a plane on a solid model while creating objects.

When in a draw command, you align the UCS by moving your pointer over an edge of a face rather than having to use the UCS command. After you finish the command, the UCS returns to its previous location and orientation.

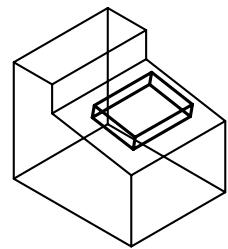
For example, you can use the dynamic UCS to create a rectangle on an angled face of a solid model as shown in the illustration.



face selected



base point and origin
of dynamic ucs



result

In the illustration on the left, the UCS is not aligned with the angled face. Instead of relocating the UCS, you turn on the dynamic UCS on the status bar or by pressing F6.



When you move the pointer completely over an edge as shown in the middle illustration, the cursor changes to show the direction of the dynamic UCS axes. You can then create objects on the angled face easily as shown in the illustration on the right.

NOTE To display the *XYZ* labels on the cursor, right-click the DUCS button and click *Display Crosshair Labels*.

The X axis of the dynamic UCS is located along an edge of the face and the positive direction of the X axis always points toward the right half of the screen. Only the front faces of a solid are detected by the dynamic UCS.

The types of commands that can use a dynamic UCS include the following:

- **Simple geometry.** Line, polyline, rectangle, arc, circle
- **Text.** Text, Multiline text, table
- **References.** Insert, xref
- **Solids.** Primitives and POLYSOLID
- **Editing.** Rotate, mirror, align
- **Other.** UCS, area, grip tool manipulation

TIP You can easily align the UCS with a plane on a solid model by turning on the dynamic UCS feature and then using the UCS command to locate the origin on that plane.

If Grid and Snap mode are turned on, they align temporarily to the dynamic UCS. The limits of the grid display are set automatically.

You can temporarily turn off the dynamic UCS by pressing F6 or SHIFT+Z while moving the pointer over a face.

NOTE The dynamic UCS is available only while a command is active.

To change the UCS dynamically

- 1 Start a command supported by dynamic UCS.
- 2 If necessary, click DUCS on the status bar to turn it on.
- 3 Move your pointer over the edge of a face on a solid model.
- 4 Complete the command.

Assign User Coordinate System Orientations to Viewports

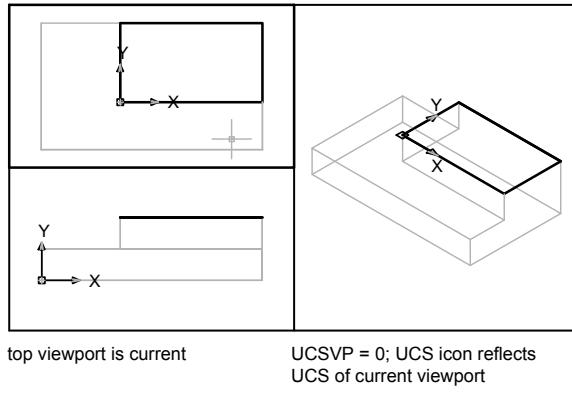
To facilitate editing objects in different views, you can define a different user coordinate system orientation for each view.

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.

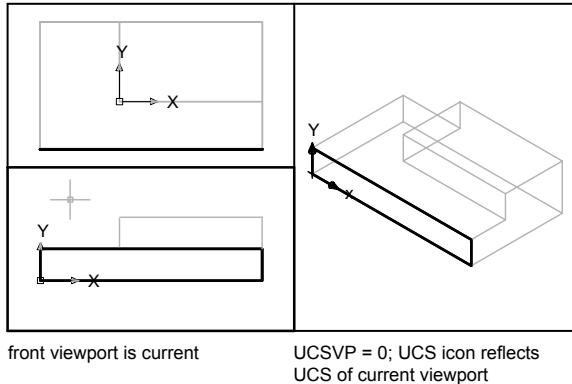
The UCS in each viewport is controlled by the UCSVP system variable. When UCSVP is set to 1 in a viewport, the UCS last used in that viewport is saved with the viewport and is restored when the viewport is made current again. When UCSVP is set to 0 in a viewport, its UCS is always the same as the UCS 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.



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.



In previous releases, the UCS was a global setting for all viewports in either model or paper space. If you want to restore the behavior of previous releases, you can set the value of the UCSVP system variable to 0 in all active viewports.

To apply the current UCS to other viewports

- 1 Make sure the UCS that you want to apply to another viewport is current.
- 2 Click Tools menu ► New UCS ► Apply.
- 3 At the prompt, click in a viewport to apply the current UCS to that viewport, or enter **all** to apply the current UCS setting to all active viewports.

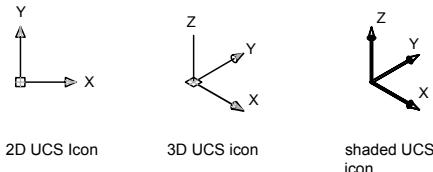
Command line: UCS

Control the Display of the User Coordinate System Icon

To help visualize the current orientation of the user coordinate system, you can display the user coordinate system icon. Several versions of this icon are available, and you can change its size, location, and color.

To indicate the location and orientation of the UCS, the UCS icon is displayed either at the UCS origin point or in the lower-left corner of the current viewport.

You can choose one of three styles of icon to represent the UCS.

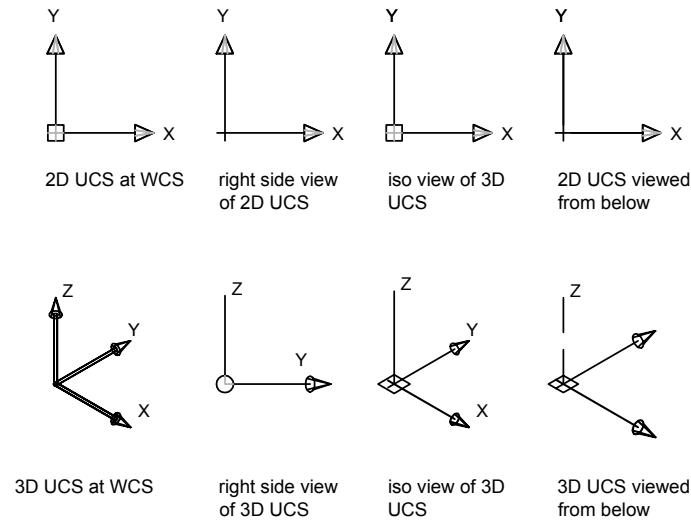


Use the UCSICON command to choose between displaying the 2D or the 3D UCS icon. The shaded UCS icon is displayed for a shaded 3D view. To indicate the origin and orientation of the UCS, you can display the UCS icon at the UCS origin point using the UCSICON command.

If the icon is displayed at the origin of the current UCS, a cross (+) appears in the icon. If the icon is displayed in the lower-left corner of the viewport, no cross appears in the icon.

If you have multiple viewports, each viewport displays its own UCS icon.

The UCS icon is displayed in various ways to help you visualize the orientation of the workplane. The following figure shows some of the possible icon displays.



You can use the **UCSICON** command to switch between the 2D UCS icon and the 3D UCS icon. You can also use the command to change the size, color, arrowhead type, and icon line width of the 3D UCS icon.

The UCS broken pencil icon replaces the 2D UCS icon when the viewing direction is in a plane parallel to the UCS XY plane. The broken pencil icon indicates that the edge of the XY plane is almost perpendicular to your viewing direction. This icon warns you not to use your pointing device to specify coordinates.

When you use the pointing device to locate a point, it's normally placed on the XY plane. If the UCS is rotated so that the Z axis lies in a plane parallel to the viewing plane—that is, if the XY plane is edge-on to the viewer—it may be difficult to visualize where the point will be located. In this case, the point will be located on a plane parallel to your viewing plane that also contains the UCS origin point. For example, if the viewing direction is along the X axis, coordinates specified with a pointing device will be located on the YZ plane, which contains the UCS origin point.

Use the 3D UCS icon to help you visualize which plane these coordinates will be projected on; the 3D UCS icon does not use a broken pencil icon.

To turn the display of the UCS icon on and off

- 1 Click View menu ► Display ► UCS Icon ► Select On.

The check mark indicates whether the icon is on or off.

Command line: UCSICON

To display the UCS icon at the UCS origin

- 1 Select View menu ► Display ► UCS Icon ► Origin.

The UCS icon is displayed at the origin of the current coordinate system.

The check mark indicates whether the option is on or off.

Command line: UCSICON

To change the appearance of the UCS icon

- 1 Click View menu ► Display ► UCS Icon ► Properties.
- 2 In the UCS Icon dialog box, change the settings.
- 3 Click OK.

Command line: UCSICON

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. Alternatively, 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 line. The difference is that your attention can stay near the cursor.

Dynamic Input is not designed to replace the command window. You can hide the command window to add screen area for drawing, but you will need

to display it for some operations. Press F2 to hide and display command prompts and error messages as needed. Alternatively, you can undock the command window and use Auto-hide to roll open or roll up the window.

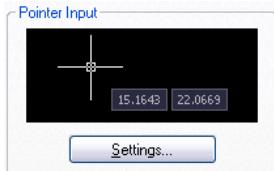
Turn Dynamic Input On and Off

Click Dyn on the status bar to turn Dynamic Input on and off. You can turn it off temporarily by holding down the F12 key. Dynamic Input has three components: pointer input, dimensional input, and dynamic prompts. Right-click Dyn and click Settings to control what is displayed by each component when Dynamic Input is on.

Pointer Input

When pointer input is on and a command is active, the location of the crosshairs is displayed as coordinates in a tooltip near the cursor. You can enter coordinate values in the tooltip instead of on the command line.

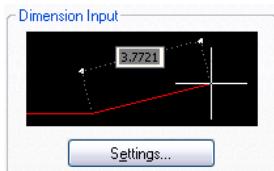
The default for second and subsequent points is relative polar coordinates (relative Cartesian for RECTANG). There is no need to type the at sign (@). If you want to use absolute coordinates, use the pound sign (#) prefix. For example, to move an object to the origin, for the second point prompt, enter #0,0.



Use the pointer input settings to change the default format for coordinates and to control when pointer input tooltips are displayed.

Dimensional Input

When dimensional input is on, the tooltips display distance and angle values when a command prompts for a second point. The values in the dimensional tooltips change as you move the cursor. Press TAB to move to the value you want to change. Dimensional input is available for ARC, CIRCLE, ELLIPSE, LINE, and PLINE.



When you use grips to edit an object, the dimensional input tooltips can display the following information:

Original length

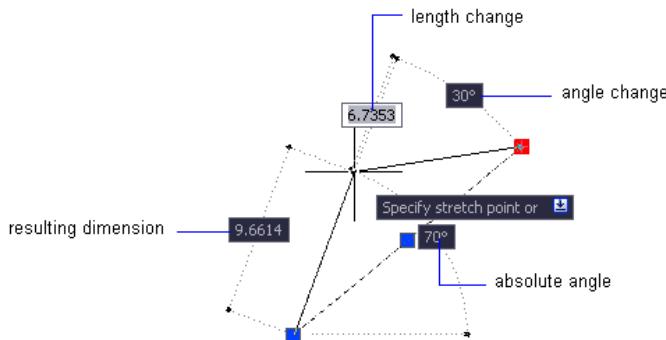
A length that updates as you move the grip

The change in the length

Angle

The change in the angle as you move the grip

The radius of an arc



Use the dimensional input settings to display only the information you want to see.

When you use grips to stretch objects or when you create new objects, dimensional input displays only acute angles, that is, all angles are displayed as 180 degrees or less. Thus, an angle of 270 degrees is displayed as 90 degrees regardless of the ANGDIR system variable setting (set in the Drawing Units dialog box). Angles specified when creating new objects rely on the cursor location to determine the positive angle direction.

Dynamic Prompts

When dynamic prompts are on, prompts are displayed in a tooltip near the cursor. You can enter a response in the tooltip instead of on the command line. Press the DOWN ARROW key to view and select options. Press the UP ARROW key to display recent input.



NOTE To use PASTECLIP in a dynamic prompt tooltip, type a letter and then backspace to delete it before you paste the entry. Otherwise, the entry is pasted into the drawing as text.

To enter coordinate values in dynamic input tooltips

- 1 On the status bar, verify that Dynamic Input (the Dyn button) is on.
- 2 Use one of the following methods to enter coordinate values or select options:
 - To enter polar coordinates, enter the distance from the first point and press TAB, and then enter an angle value and press ENTER.
 - To enter Cartesian coordinates, enter an *X* coordinate value and a comma (,), and then enter a *Y* coordinate value and press ENTER.
 - If a down-arrow icon follows the prompt, press the DOWN ARROW key until a dot is displayed next to the option. Press ENTER.
 - Press the UP ARROW key to access recent coordinates, or right-click and click Recent Input to access the coordinates from a shortcut menu.

NOTE For dimensional input, after you type a value in an input field and press TAB, the field then displays a lock icon, and the cursor is constrained by the value that you entered.

To correct typing errors in dynamic input tooltips

- When a dynamic input tooltip displays the red error outline, the current entry is selected. Type over the selected text to replace it. You can also use the RIGHT ARROW, LEFT ARROW, BACKSPACE, and DELETE keys to correct your entry. After you make the correction, press TAB, comma (,), or a left angle bracket (<) to remove the red outline and complete the coordinates.
- If you type the @ or # or * prefixes in a pointer input tooltip and then want to change it, you can just type the character you want. There is no need to backspace.

To specify relative or absolute coordinates in pointer input tooltips

- To enter absolute coordinates when relative coordinates are displayed in the tooltip, enter # to temporarily override the DYNPICOORDS system variable.

- To enter relative coordinates when absolute coordinates are displayed, enter @ to temporarily override the DYNPICOORDS system variable.
- To enter absolute world coordinate system (WCS) coordinates, enter *.

NOTE During pointer input, you can use the shortcut menu to access the # and * prefixes.

To choose options in dynamic prompt tooltips

- 1 Start a command.

Tooltips near the crosshairs display the coordinates of the cursor location and a prompt.

- 2 If a prompt is displayed, press the DOWN ARROW key to display the options.
- 3 Press DOWN ARROW or UP ARROW to place a dot next to the option you want. Press ENTER.
Press ESC to undo the most recent action.
- 4 Specify points and enter options to complete the command.

To change the color, size, or transparency of tooltips

- 1 Right-click Dyn on the status bar. Click Settings.
- 2 In the Drafting Settings dialog box, Dynamic input tab, click Drafting Tooltip Appearance.
- 3 In the Tooltip Appearance dialog box, under Color, click Model Color or Layout Color to display the “Select Color Dialog Box”, where you can specify a color for tooltips in the space you selected.
- 4 Under Size, move the slider to the right to make tooltips larger or to the left to make them smaller. The default value, 0, is in the middle.
- 5 Under Transparency, move the slider. The lower the setting, the more transparent the tooltip. A value of 100 sets the tooltip to opaque.
- 6 Under Apply To, choose an option:
 - **Override OS Settings for All Drafting Tooltips.** Applies the settings to all tooltips, overriding the settings in the operating system.
 - **Use Settings Only for Dynamic Input Tooltips.** Applies the settings only to the drafting tooltips used in Dynamic Input.
- 7 Click OK.

Command line: DYNTOOLTIPS

To merge the information in all drafting tooltips into one tooltip

- On the command line, enter **tooltipmerge**.

Tooltips that display drafting information are merged into one.

To turn on or turn off Dynamic Input

- On the status bar, click Dyn, or press F12.

To turn off Dynamic Input temporarily, hold down the F12 key while you work.

NOTE The F12 temporary override key does not turn off Dynamic Input.

To change pointer input settings

- 1 Right-click Dyn on the status bar. Click Settings.
- 2 In the Drafting Settings dialog box, Dynamic input tab, under Pointer Input, click Settings.
- 3 In the Pointer Input Settings dialog box, select polar or Cartesian format as the default.
- 4 Select relative or absolute coordinate format as the default.
- 5 Under Visibility, select one of the following options:
 - **As Soon As I Type Coordinate Data.** When pointer input is turned on, displays tooltips only when you start to enter coordinate data.
 - **When a Command Asks for a Point.** When pointer input is turned on, displays tooltips whenever a command prompts for a point.
 - **Always—Even When Not in a Command.** Always displays tooltips when pointer input is turned on.
- 6 Click OK to close each dialog box.

Command line: DSETTINGS

To change dimensional input settings

- 1 Right-click Dyn on the status bar. Click Settings.

- 2 In the Drafting Settings dialog box, Dynamic input tab, under Dimension Input, click Settings.
- 3 In the Dimension Input Settings dialog box, select Polar or Cartesian format as the default.
- 4 Under Visibility, select one of the following options:
 - **Show Only 1 Dimension Input Field at a Time.** Displays only the distance dimensional input tooltip when you are using grip editing to stretch an object.
 - **Show 2 Dimension Input Fields at a Time.** Displays the distance and angle dimensional input tooltips when you are using grip editing to stretch an object.
 - **Show the Following Dimension Input Fields Simultaneously.** Displays the selected dimensional input tooltips when you are using grip editing to stretch an object. Select one or more of the check boxes.
- 5 Click OK to close each dialog box.

Command line: DSETTINGS

To display prompts in tooltips

- 1 Right-click Dyn on the status bar. Click Settings.
- 2 In the Drafting Settings dialog box, Dynamic input tab, under Dynamic Prompts, check Show Command Prompting and Command Input Near the Crosshairs.
- 3 Click OK.

Command line: DSETTINGS

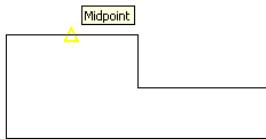
Snap to Locations on Objects (Object Snaps)

Instead of entering coordinates, you can specify points relative to existing objects such as endpoints of lines or center points of circles.

Use Object Snaps

Use object snaps to specify precise locations on objects. For example, you can use an object snap to draw a line to the center of a circle or to the midpoint of a polyline segment.

You can specify an object snap whenever you are prompted for a point. By default, a marker and a tooltip are displayed when you move the cursor over an object snap location on an object. This feature, called AutoSnap™, provides a visual clue that indicates which object snaps are in effect.



For a list of object snaps, see OSNAP.

Specify an Object Snap

To specify an object snap at a prompt for a point, you can

- Press SHIFT and right-click to display the Object Snap shortcut menu
- Click an object snap button on the Object Snap toolbar
- Enter the name of an object snap on the command line

When you specify an object snap at a prompt for a point, the object snap stays in effect only for the next point that you specify.

NOTE Object snaps work only when you are prompted for a point. If you try to use an object snap at the Command prompt, an error message is displayed.

Use Running Object Snaps

If you need to use one or more object snaps repeatedly, you can turn on *running object snaps*. For example, you might set Center as a running object snap if you need to connect the centers of a series of circles with a line.

You can specify one or more running object snaps on the Object Snaps tab in the Drafting Settings dialog box, which is accessible from the Tools menu. If several running object snaps are on, more than one object snap may be eligible at a given location. Press TAB to cycle through the possibilities before you specify the point.

Click the OSNAP button on the status bar or press F3 to turn running object snaps on and off.

NOTE If you want object snaps to ignore hatch objects, set the OSOPTIONS system variable to 1.

Use Object Snaps in 3D

By default, the Z-value of an object snap location is determined by the object's location in space. However, if you work with object snaps on the plan view of a building or the top view of a part, a constant Z-value is more useful.

If you turn on the OSNAPZ system variable, all object snaps are projected onto the XY plane of the current UCS or, if ELEV is set to a non-zero value, onto a plane parallel to XY plane at the specified elevation.

NOTE When you draw or modify objects, make sure that you know whether OSNAPZ is on or off. There is no visual reminder, and you can get unexpected results.

To snap to a geometric point on an object

- 1 At the prompt for a point, hold down SHIFT and right-click in the drawing area. Select the object snap you want to use.
- 2 Move your cursor over the desired object snap location.
If AutoSnap is on, your cursor automatically locks onto the snap location you selected, and a marker and tooltip indicate the object snap point.
- 3 Select an object.
The cursor snaps to the eligible location closest to your selection.

To set running object snaps

- 1 Click Tools menu ► Drafting Settings.
- 2 In the Drafting Settings dialog box, Object Snap tab, select the object snaps you want to use.
- 3 Click OK.

Object Snap toolbar



Command line: OSNAP

Shortcut menu: Hold down SHIFT and right-click to display the object snap menu. Click Osnap Settings.

To turn on and turn off running object snaps as you work

- On the status bar, click Osnap, or press F3.

If running object snaps have been set, the settings are turned on or off.

To turn running object snaps on and off temporarily, hold down the F3 key while you work.

Command line: OSNAP

To set object snaps to ignore hatch objects

- 1 At the Command prompt, enter **osoptions**.
- 2 Enter **1**.

The Object Snap Menu

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

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

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

See also:

- ["Create Pull-Down and Shortcut Menus" in the *Customization Guide*](#)
[Use Object Snaps](#)

To display the object snap menu

- 1 Enter any command that prompts you to specify a point. For example, enter **line**.
- 2 At the From Point prompt, hold down SHIFT and right-click.
The object snap menu is displayed, and you can click an object snap option.

Set Visual Aids for Object Snaps (AutoSnap)

Object snaps include a visual aid called AutoSnap™ to help you see and use object snaps more efficiently. AutoSnap displays a marker and a tooltip when you move your cursor over an object snap location.

AutoSnap Tools

AutoSnap consists of the following snap tools:

- **Marker.** Displays the object snap location when the cursor moves over or near an object. Marker shape is dependent on the snap it is marking.
- **Tooltip.** Describes which part of the object you are snapping to in a small flag at the cursor location.
- **Magnet.** Attracts and locks the cursor onto the nearest detected snap points. Provides a visual cue, similar to snapping to a grid.
- **Aperture box.** Surrounds the crosshairs and defines an area within which object snaps are evaluated. You can choose to display or not display the aperture box, and you can change the aperture box size.

The AutoSnap markers, tooltips, and magnet are turned on by default. You can change AutoSnap settings on the Drafting tab in the Options dialog box.

Use AutoSnap to Confirm or Change an Object Snap

If you have set more than one running object snap, you can press TAB to cycle through all the object snap points available for a particular object.

To change the AutoSnap settings

- 1 Click Tools menu ▶ Options.
- 2 In the Options dialog box, Drafting tab, change settings as needed:
 - **Marker.** Turns the marker on or off.
 - **Magnet.** Turns the magnet on or off.
 - **Display AutoSnap Tooltip.** Turns tooltips on or off.
 - **Display AutoSnap Aperture Box.** Turns the target box on or off when you specify an object snap. This setting does not affect object snaps when you are not using AutoSnap.
 - **AutoSnap Marker Color.** Changes the color of the marker.
 - **AutoSnap Marker Size.** Adjusts the size of the marker.

- 3 Click OK.

Object Snap toolbar



Command line: OPTIONS

Override Object Snap Settings

While you work, you can turn running object snaps on and off temporarily by using an override key. Temporary override keys can also be used for other drawing aids; for example, Ortho mode and Polar mode.

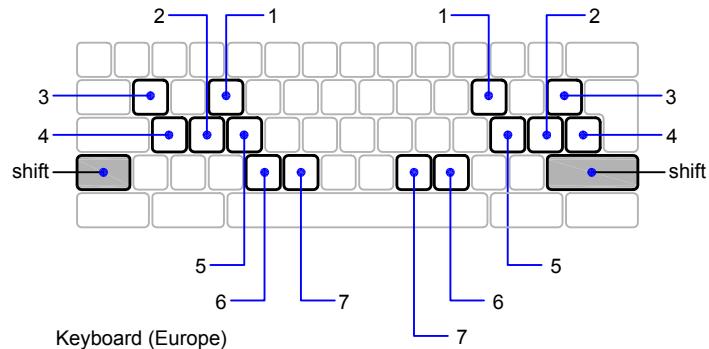
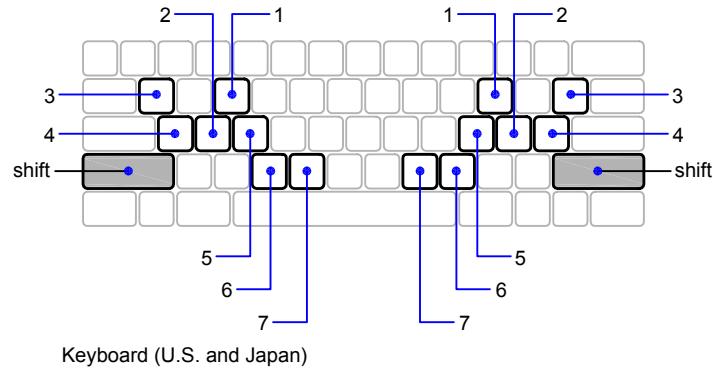
For example, if you have set running object snaps but you want to turn them off for one point, you can hold down F3. When you release this override key, running object snaps are restored.

There are also temporary override keys for individual object snaps. Override keys are set up to be easy to find by touch without looking away from your drawing.

The keys in the following illustration are the default keys, but you can change key assignments and add your own as needed.

Hold down SHIFT and one of the temporary override keys in the illustration:

- | | |
|------------------------------------|---------------------------------------|
| 1 Object snap override: Endpoint | 5 Turns off all snapping and tracking |
| 2 Enforces object snap selection | 6 Object snap override: Center |
| 3 Toggles object snap tracking | 7 Object snap override: Midpoint |
| 4 Toggles object snap mode (OSNAP) | |
-



Temporary override keys are also available for the other drawing aids that you set in the “Drafting Settings Dialog Box”.

See also:

- “Adjust Grid and Grid Snap”
- “Use Orthogonal Locking (Ortho Mode)”
- “Use Polar Tracking and PolarSnap”
- “Use Dynamic Input”
- Add Shortcut Keys and Temporary Override Keys in the *Customization Guide*

To temporarily override the running object snap settings

- Hold down F3 while you work.

When you release the key, the current running object snap settings are restored.

To change the keyboard response time for temporary override keys

- 1 In the Windows Control Panel, click Keyboard.
- 2 In the Keyboard Properties dialog box, on the Speed tab, drag the Repeat Rate slider to adjust the keyboard response time. Then click OK.

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

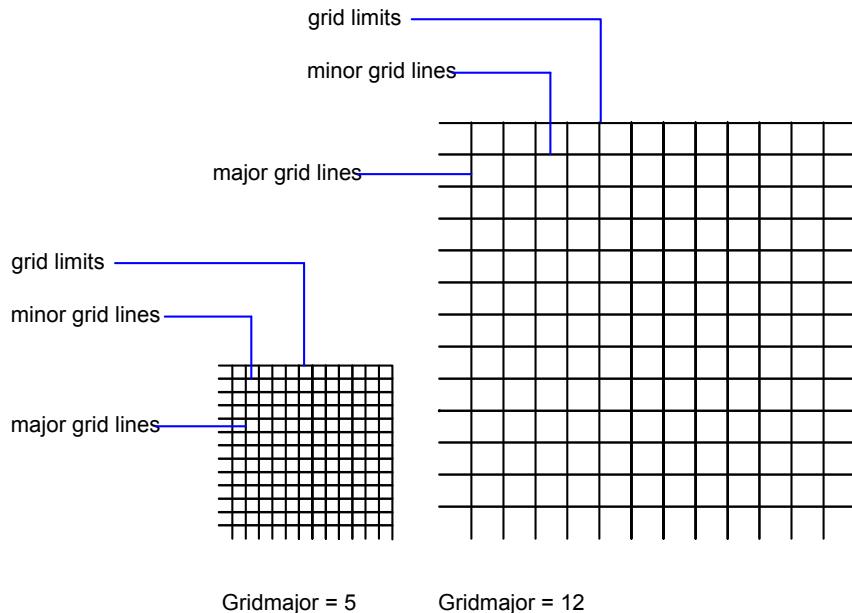
By default, the X and Y axes of the UCS display in a different color than the grid lines. You can control the color in the Drawing Window Colors dialog box. This dialog box is accessible from the Drafting tab in the Options dialog box.

The LIMITS command controls the drawing area covered by the grid. As an option, you can override the limits to make the grid cover the entire XY plane of the user coordinate system (UCS). You can access this option in the Drafting Settings dialog box or use the GRIDDISPLAY system variable.

NOTE When you use dynamic UCS, the grid limits are set automatically relative to the size of the selected face of the solid and the drawing area available.

Control the Frequency of Major Grid Lines

If the grid is displayed as lines rather than dots, darker lines called *major grid lines* display at intervals. When working in decimal units or with feet and inches, major grid lines are especially useful for measuring distances quickly. You can control the frequency of major grid lines in the Drafting Settings dialog box.



To turn off the display of major grid lines, set the frequency of major grid lines to 1.

NOTE If the grid is displayed as lines, the grid limits are displayed also as darker lines. Do not confuse these boundaries with major grid lines.

NOTE When the grid is displayed as lines and SNAPANG is set to a value other than 0, the grid will not display. SNAPANG does not affect the display of the dotted grid.

Change the Grid Dynamically During Zooming

If you zoom in or out of your drawing, the grid spacing is adjusted automatically to be more appropriate for the new magnification. This is called *adaptive grid display*.

For example, if you zoom way out, the density of displayed grid lines reduces automatically. Conversely, if you zoom way in, additional grid lines display in the same proportion as the major grid lines.

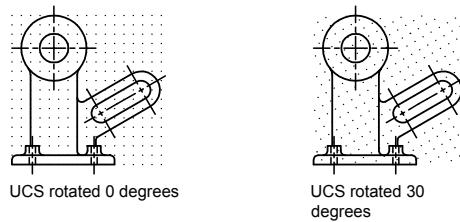
Change Grid and Snap Spacing

As you work, you can turn Grid and Snap mode on and off, and you can change the grid and snap spacing. You can turn Snap mode on and off temporarily by using an override key.

Snap spacing does not have to match grid spacing. For example, you might set a wide grid spacing to be used as a reference but maintain a closer snap spacing for accuracy in specifying points.

Change the Grid and Snap Angle and Base

If you need to draw along a specific alignment or angle, you can change the grid and snap angle by rotating the user coordinate system (UCS). This rotation realigns the crosshairs on the screen to match the new angle. In the following example, the UCS is rotated 30 degrees to match the angle of the anchor bracket.



The grid and snap points are always aligned with the UCS origin. If you need to shift the grid and grid snap origin, move the UCS.

See also:

- ["Set Isometric Grid and Snap"](#)
- [Override Object Snap Settings](#)
- [Use a Visual Style to Display Your Model](#)

To display a grid and set grid spacing

- 1 Click Tools menu ► Drafting Settings.
- 2 In the Drafting Settings dialog box, Snap and Grid tab, select Grid On to display the grid.
- 3 Under Snap Type, make sure Grid Snap and Rectangular Snap are selected.
- 4 For Grid X Spacing, enter the horizontal grid spacing in units.
- 5 To use the same value for vertical grid spacing, press ENTER. Otherwise, enter a new value for Grid Y Spacing.
- 6 Click OK.

Command line: DSETTINGS

To turn on Snap mode and set snap spacing

- 1 Click Tools menu ► Drafting Settings.
- 2 In the Drafting Settings dialog box, Snap and Grid tab, select Snap On.
- 3 Under Snap Type, make sure Grid Snap and Rectangular Snap are selected.
- 4 In the Snap X Spacing box, enter the horizontal snap spacing value in units.
- 5 To specify the same vertical snap spacing, press ENTER. Otherwise, enter a new distance in the Snap Y Spacing box.
- 6 Click OK.

Command line: DSETTINGS

To set the grid limits

- 1 Click Format menu ► Drawing Limits.
- 2 At the Command prompt, enter the coordinate values for a point at the lower left corner of the grid limits.
- 3 Enter the coordinate values for a point at the upper right corner of the grid limits.

The grid limits are set to a rectangular area defined by the two points.

Command line: LIMITS

To temporarily override Snap mode

- Hold down F9 while you work.
When you release the key, Snap mode is restored.

To rotate the grid and snap angle and change the base point

- 1 Click Tools menu ▶ New UCS ▶ Z.
- 2 Enter the rotation angle for the UCS.
- 3 Click Tools menu ▶ New UCS ▶ Origin.
- 4 Specify a new origin point for the UCS.
- 5 Click OK.

Command line: UCS

To change the grid display between dots and lines

- 1 At the Command prompt, enter VSCURRENT.
- 2 Do *one* of the following:
 - To display the grid as dots, specify the 2D Wireframe option.
 - To display the grid as lines, specify any other option.

Command line: VSCURRENT

To change the frequency of major grid lines

- 1 If necessary, at the Command prompt, enter VSCURRENT and specify any visual style other than 2D Wireframe.
- 2 Click Tools menu » Drafting Settings.
- 3 In the Drafting Settings dialog box, Snap and Grid tab, specify a number for Major Line Every.
- 4 Click OK.

Command line: VSCURRENT

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 377.

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](#)

To turn on or turn off Ortho mode

- On the status bar, click Ortho.

To turn Ortho on or off temporarily, hold down the SHIFT key while you work. While you use the temporary override key, the direct distance entry method is not available.

NOTE Turning on Ortho automatically turns off polar tracking.



Command line: ORTHO

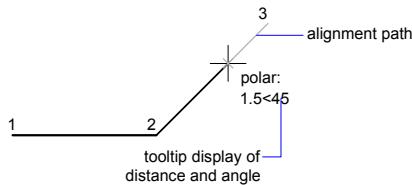
Use Polar Tracking and PolarSnap

Polar tracking restricts cursor movement to specified angles. PolarSnap restricts cursor movement to specified increments along a polar angle.

When you are creating or modifying objects, you can use polar tracking to display temporary alignment paths defined by the polar angles you specify. In 3D views, polar tracking additionally provides an alignment path in the up and down directions. In that case, the tooltip displays a +Z or -Z for the angle.

Polar angles are relative to the orientation of the current user coordinate system (UCS) and the setting for the base angle convention in a drawing. The angle base direction is set in the Drawing Units dialog box.

Use PolarSnap™ to snap to specified distances along the alignment path. For example, in the following illustration you draw a two-unit line from point 1 to point 2, and then draw a two-unit line to point 3 at a 45-degree angle to the line. If you turn on the 45-degree polar angle increment, an alignment path and tooltip are displayed when your cursor crosses the 0 or 45-degree angle. The alignment path and tooltip disappear when you move the cursor away from the angle.

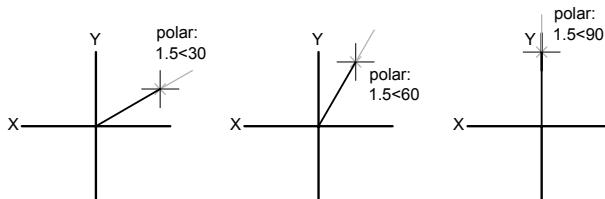


As you move your cursor, alignment paths and tooltips are displayed when you move the cursor near polar angles. The default angle measurement is 90 degrees. Use the alignment path and tooltip to draw your object. You can use polar tracking with Intersection and Apparent Intersection object snaps to find where a polar alignment path intersects another object.

NOTE Ortho mode and polar tracking cannot be on at the same time. Turning on polar tracking turns off Ortho mode. Similarly, PolarSnap and grid snap cannot be on at the same time. Turning on PolarSnap turns off grid snap.

Specify Polar Angles (Polar Tracking)

You can use polar tracking to track along polar angle increments of 90, 60, 45, 30, 22.5, 18, 15, 10, and 5 degrees, or you can specify other angles. The following illustration shows the alignment paths displayed as you move your cursor 90 degrees with the polar angle increment set to 30 degrees.



The orientation of 0 depends on the angle you set in the Drawing Units dialog box (UNITS). The direction of snap (clockwise or counterclockwise) depends on the units direction you specify when setting units of measurement.

You can turn polar tracking on and off temporarily by using an override key. The direct distance entry method is not available while you are using the temporary override key for polar tracking.

Specify Polar Distances (PolarSnap)

PolarSnap restricts cursor movement to increments of a polar distance you specify. For example, if you specify a length of 4 units, the cursor snaps from the first point specified to lengths of 0, 4, 8, 12, 16, and so on. As you move your cursor, a tooltip indicates the nearest PolarSnap increment. To restrict point entry to polar distances, both polar tracking and Snap mode (set to PolarSnap) must be on. You can turn off all snapping and tracking temporarily by using an override key.

See also:

[Override Object Snap Settings](#)

To turn on and turn off polar tracking

- Press F10, or click Polar on the status bar.

To turn polar tracking on or off temporarily, hold down the F10 key while you work.

To set polar snap distance

- 1 Click Tools menu ▶ Drafting Settings.
- 2 In the Drafting Settings dialog box, Snap and Grid tab, select Snap On.
- 3 In Snap Type, select PolarSnap.
- 4 Under Polar Spacing, enter the polar distance.
- 5 On the Polar Tracking tab, select Polar Tracking On.
- 6 Select the angle from the Increment Angle list.
You can specify your own angles by choosing Additional Angles and then New.
- 7 Click OK.

Command line: DSETTINGS

To draw objects using polar tracking

- 1 Turn on polar tracking and start a drawing command, such as ARC, CIRCLE, or LINE.
You can also use polar tracking with editing commands, such as COPY and MOVE.
- 2 As you move your cursor to specify points, notice the dotted polar tracking line that appears at the tracking angles you specified. Points you specify while the line is displayed conform to the polar tracking angle.

Command line: DSETTINGS

To draw objects using polar distance

- 1 Turn on snap and polar tracking.
Make sure Polar Snap is selected in the Drafting Settings dialog box, Snap & Grid tab.
- 2 Start a drawing command, such as LINE.
- 3 As you move your cursor, notice that the dotted polar tracking line displays a tooltip that shows distance and angle.
- 4 Specify a point.

The length of the new line conforms to the polar distance.

Command line: DSETTINGS

To set polar tracking angles

- 1 Click Tools menu ► Drafting Settings.
- 2 In the Drafting Settings dialog box, Polar Tracking tab, select Polar Tracking On.
- 3 In the Increment Angle list, select the polar tracking angle.
- 4 To set additional tracking angles, select Additional Angles. Click New. Enter the angle value in the text box.
- 5 Under Polar Angle Measurement, specify whether polar tracking increments are based on the UCS or relative to the last object you created.
- 6 Click OK.

Command line: DSETTINGS

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.

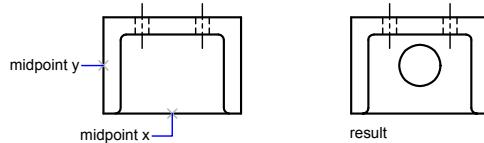
Coordinate filters specify a new coordinate location by using the *X* value from one location, the *Y* value of a second location, and, for 3D coordinates, the *Z* value of a third location. When used with object snaps, coordinate filters extract coordinate values from an existing object.

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 on the command line, enter a period and one or more of the letters *X*, *Y*, and *Z*. The next entry is limited to a specific coordinate value.

Example: Use of Coordinate Filters in 2D

In the following illustration, the hole in the holding plate was centered in the rectangle by extracting the *X*,*Y* coordinates from the midpoints of the plate's horizontal and vertical line segments.



Here is the command line sequence:

Command: **circle**

Specify center point for circle or [3P/2P/Ttr (tangent tangent radius)]: .x

of: **mid**

of: Select the horizontal line on the lower edge of the holding plate

of: (need YZ): **mid**

of: Select the vertical line on the left side of the holding plate

of: Diameter/<Radius> Specify the radius of the hole

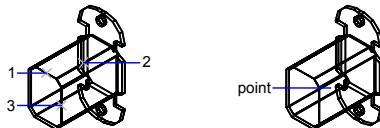
Coordinate filters work only when the program prompts you for a point. If you try to use a coordinate filter at the Command prompt, you see an error message.

Example: Use of Coordinate Filters in 3D

This example shows how to use coordinate filters to create a point object at the center (centroid) of a 3D object. Hidden lines have been removed for clarity. The X value of the new point is extracted from the first location specified, the Y value from the second location, and the Z value from the third. The three values are combined to form the coordinate values of the new point.

Command: **point**

```
Point: .x  
of mid  
of select object (1)  
(need YZ): .y  
of mid  
of select object (2)  
(need Z): mid  
of select object (3)
```



To use coordinate filters to specify a point in 2D

- 1 At the prompt for a point, enter a coordinate filter (.x or .y).

For example, enter .x to specify the X value first.

- 2 To extract the first coordinate value, specify a point.

For example, if you entered .x in step 1, the X value is extracted from this point.

- 3 To extract the next coordinate value, specify a different point.

The new point location combines the coordinate values extracted from the points you specified in steps 2 and 3.

NOTE Instead of specifying a point in steps 2 or 3, you can enter a numeric value.

To use coordinate filters to specify a point in 3D

- 1 At the prompt for a point, enter a coordinate filter (.x, .y, .z, .xy, .xz, or .yz).

For example, enter **.x** to specify the *X* value first.

- 2 To extract the specified coordinate value(s), specify a point.

For example, if you entered **.x** in step 1, the *X* value is extracted from this point.

- 3 At the prompt for the remaining coordinates, do one of the following:

- Extract the remaining coordinate values by specifying a point.

- Enter another coordinate filter and return to step 2.

For example, if you entered **.x** in step 1, specify a second point to extract the *Y* and *Z* coordinates simultaneously, or enter **.y** or **.z** to specify *Y* and *Z* values separately.

The new point location combines the coordinate values extracted from the points specified in steps 2 and 3.

NOTE Instead of specifying a point in steps 2 or 3, you can enter a numeric value.

Track to Points on Objects (Object Snap Tracking)

You can draw objects at specific angles or in specific relationship to other objects along specified directions called alignment paths.

AutoTrack™ helps you draw objects at specific angles or in specific relationships to other objects. When you turn on AutoTrack, temporary *alignment* paths help you create objects at precise positions and angles. AutoTrack includes two tracking options: polar tracking and object snap tracking.

You can toggle AutoTrack on and off with the Polar and Otrack buttons on the status bar. Use temporary override keys to turn object snap tracking on and off or to turn off all snapping and tracking. See the keyboard illustration in Override Object Snap Settings.

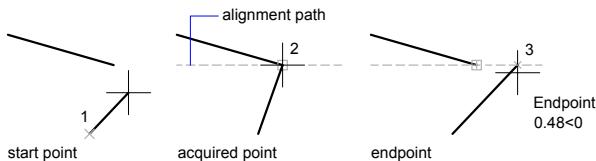
Object snap tracking works in conjunction with object snaps. You must set an object snap before you can track from an object's snap point.

Object Snap Tracking

Use object snap tracking to track along alignment paths that are based on object snap points. Acquired points display a small plus sign (+), and you can acquire up to seven tracking points at a time. After you acquire a point, horizontal, vertical, or polar alignment paths relative to the point are displayed as you move the cursor over their drawing paths. For example, you can select

a point along a path based on an object endpoint or midpoint or an intersection between objects.

In the following illustration, the Endpoint object snap is on. You start a line by clicking its start point (1), move the cursor over another line's endpoint (2) to acquire it, and then move the cursor along the horizontal alignment path to locate the endpoint you want for the line you are drawing (3).



Change Object Snap Tracking Settings

By default, object snap tracking is set to orthogonal. Alignment paths are displayed at 0, 90, 180, and 270 degrees from acquired object points. However, you can use polar tracking angles instead.

For object snap tracking, object points are automatically acquired. However, you can choose to acquire points only when you press SHIFT.

Change Alignment Path Display

You can change how AutoTrack displays alignment paths, and you can change how object points are acquired for object snap tracking. By default, alignment paths stretch to the end of the drawing window. You can change their display to abbreviated lengths, or no length.

Tips for Using Object Snap Tracking

As you use AutoTrack (polar tracking and object snap tracking), you will discover techniques that make specific design tasks easier. Here are a few you might try.

- Use Perpendicular, End, and Mid object snaps with object snap tracking to draw to points that are perpendicular to the end and midpoints of objects.
- Use the Tangent and End object snaps with object snap tracking to draw to points that are tangent to the endpoints of arcs.
- Use object snap tracking with temporary tracking points. At a point prompt, enter **tt**, then specify a temporary tracking point. A small + appears at the point. As you move your cursor, AutoTrack alignment paths are displayed

relative to the temporary point. To remove the point, move the cursor back over the +.

- After you acquire an object snap point, use direct distance to specify points at precise distances along alignment paths from the acquired object snap point. To specify a point prompt, select an object snap, move the cursor to display an alignment path, then enter a distance at the Command prompt.

NOTE The direct distance entry method is not available while you are using the temporary override key for object snap tracking.

- Use the Automatic and SHIFT to Acquire options set on the Drafting tab of the Options dialog box to manage point acquisition. Point acquisition is set to Automatic by default. When working in close quarters, press SHIFT to temporarily avoid acquiring a point.

To turn on and turn off object snap tracking

- Press F11, or click Otrack on the status bar.

To turn object snap tracking on and off temporarily, hold down the F11 key while you work.

To change AutoTrack settings

- 1 Click Tools menu ▶ Options.
- 2 In the Options dialog box, Drafting tab, under AutoTrack Settings, select or clear the following alignment path display options:
 - **Display Polar Tracking Vector.** Controls alignment path display for object snap tracking. When cleared, no polar tracking path is displayed.
 - **Display Full Screen Tracking Vector.** Controls alignment path display for object snap tracking. When cleared, an alignment path is displayed only from the object snap point to the cursor.
 - **Display AutoTrack Tooltip.** Controls the display of AutoTrack tooltips. Tooltips tell you the type of object snap (for object snap tracking), alignment angle, and distance from the previous point.
- 3 Under Alignment Point Acquisition, select a method for acquiring object points for object snap tracking:
 - **Automatic.** Acquires object points automatically. If you select this option, you can press SHIFT to not acquire an object point

- **Shift to Acquire.** Acquires object points only when you press SHIFT while the cursor is over an object snap point.

Command line: OPTIONS

Track to Offset Point Locations (Tracking)

You can use tracking to specify a point by offsetting vertically and horizontally from a series of temporary points.

You can use the tracking method whenever you are prompted for a point. Tracking uses the pointing device to specify a point by offsetting vertically and horizontally from a series of temporary points. When you start tracking and specify an initial reference point, the next reference point is constrained to a path that extends vertically or horizontally from that point. The direction of the offset is indicated by the rubber-band line. You change the direction of the offset by moving the cursor through the reference point. You can track as many points as you need. Typically, you use tracking in combination with object snaps or direct distance entry.

For example, you can use tracking to find the center point of a rectangle without using construction lines. Start tracking, and specify the midpoint of a horizontal line. Drag the cursor vertically and specify the midpoint of a vertical line (2). Press ENTER to accept the point (3) at the center of the rectangle.

To use tracking to specify a point

- 1 Start a command, such as the LINE command.
- 2 Hold down SHIFT and right-click in the drawing area. Click Tracking.
- 3 Specify a point.
- 4 Move the cursor directly up, down, left, or right until you see the rubber-band line.

The direction of movement affects the tracking direction. Notice that if you move the cursor from left to right, you must then move it directly over the last point specified in order to move it up or down.

- 5 Specify a second point.
- 6 Press ENTER to end tracking.

The start point of the line snaps to the imaginary intersection of the vertical and horizontal paths extending from the points you specified. The position is determined by the direction in which you moved the cursor after specifying the first point.

Command line: TRACKING (Command Modifier)

Specify Distances

When specifying a point, you can enter distances, offsets, and measured intervals.

Enter Direct Distances

You can specify a point by moving the cursor to indicate a direction and then entering the distance.

To specify a line length quickly, without entering coordinate values, you can specify a point by moving the cursor to indicate a direction and then entering the distance from the first point. You can enter calculated distances from the QuickCalc calculator. For more information, see Use the QuickCalc Calculator.

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 377
- “Lock an Angle for One Point (Angle)” on page 380

To draw a line using direct distance entry

- 1 Start the LINE command and specify the first point.
- 2 Move the pointing device until the rubber-band line extends at the same angle as the line you want to draw.
- 3 Enter a distance on the command line.
The line is drawn at the length and angle you specified.

Offset from Temporary Reference Points

You can establish a temporary reference point as a base point for offsetting subsequent points.

The From command modifier establishes a temporary reference point as a base point for offsetting subsequent points. The From method does not constrain the cursor to orthogonal movement. The From method usually is used in combination with object snaps.

To offset a point from a temporary reference point

- 1 At a prompt for a point, enter **from**.
- 2 If you want to offset from a location on an existing object, specify an object snap method. Then select the object.
- 3 Enter a relative coordinate.

Press SHIFT and right-click to display the object snap menu.

Object Snap toolbar



Specify Intervals on Objects

You can mark off equal distances along objects.

Overview of Specifying Intervals on Objects

Provides a high-level overview of two options for marking off equal distances along objects.

Sometimes you need to create points or insert symbols (blocks) at intervals on an object.

You can

- Specify the length of the segments (MEASURE)
- Specify the number of equal segments (DIVIDE)

You can measure or divide lines, arcs, splines, circles, ellipses, and polylines. With both methods, you can identify the intervals by inserting either a point or a block.

By specifying points, you can use the Node object snap to align other objects at intervals on the measured or divided object. By specifying blocks, you can create precise geometric constructions or insert custom markers. The blocks can rotate at each insertion point.

You cannot insert a block unless it has already been defined within the drawing. Variable attributes within the block are not included when you insert the block references.

The points or blocks you draw using MEASURE or DIVIDE are placed in a selection set. Therefore, if you want to edit them immediately, you can use the Previous option of SELECT.

See also:

["Create and Use Blocks \(Symbols\)" on page 459](#)

Specify Measured Intervals on Objects

You can mark off equal lengths from one end of a selected object.

You can use MEASURE to mark an object at specified intervals. You can mark the intervals with either points or blocks. The last segment of a measured object may be shorter than the interval you specify.

The starting point for measurements or divisions varies with the object type. For lines or open polylines, the starting point is the endpoint closest to the selection point. For closed polylines, it is the polyline start point. For circles, it is at the angle from the center point that is equivalent to the current snap angle. For example, if the snap angle is 0, the circle starts at the three o'clock position and continues counterclockwise.

If the point marker is displayed as a single dot (the default setting), you may not be able to see the measured intervals. You can change the style of the point markers using several methods. To change the point style in a dialog box, you can use DDPTYPE, or choose Point Style from the Format menu. The PDMODE system variable also controls the appearance of point markers. For example, you can change the value to make points appear as crosses. PDSIZE controls the size of point objects.

To insert points at measured intervals on an object

- 1 Click Draw menu ▶ Point ▶ Measure.
- 2 Select a line, arc, spline, circle, ellipse, or polyline.
- 3 Enter an interval length, or specify points to indicate a length.

Points are placed on the object at the specified intervals.

Command line: MEASURE

To insert blocks at measured intervals on an object

- 1 If necessary, create the block you want to insert.

- 2 Click Draw menu ► Point ► Measure.
- 3 Select a line, arc, spline, circle, ellipse, or polyline.
- 4 Enter **b** (Block).
- 5 Enter the name of the block you want to insert.
- 6 Enter **y** to align the blocks with the measured object. Enter **n** to use a rotation angle of 0 degrees.
- 7 Enter an interval length, or specify points to indicate a length.
Blocks are inserted on the object at the specified intervals.

Command line: BLOCK, MEASURE

Divide an Object into Equal Segments

You can divide a selected object into a specified number of equal lengths.

You can create points or insert blocks on an object at a specific number of equal intervals. This operation does not actually break an object into individual objects; it only identifies the location of the divisions so that you can use them as geometric reference points.



The starting point for measurements or divisions varies with the object type. For lines or open polylines, the starting point is the endpoint closest to the selection point. For closed polylines, it is the polyline start point. For circles, it is at the angle from the center point that is equivalent to the current snap angle. For example, if the snap angle is 0, the circle starts at the three o'clock position and continues counterclockwise.

If the point marker is displayed as a single dot (the default setting), you may not be able to see the segments. You can change the style of the point markers using several methods. To change the point style in a dialog box, you can use DDPTYPE, or choose Point Style from the Format menu. The PDMODE system variable also controls the appearance of point markers. For example, you can change the value to make points appear as crosses. PDSIZE controls the size of point objects.

To insert points to mark equal segments

- 1 Click Draw menu ▶ Point ▶ Divide.
- 2 Select a line, circle, ellipse, polyline, arc, or spline.
- 3 Enter the number of segments you want.
A point is placed between each segment.

Command line: DIVIDE

To insert blocks to mark equal segments on an object

- 1 If necessary, create the block you want to insert.
- 2 Click Draw menu ▶ Point ▶ Divide.
- 3 Select a line, arc, circle, ellipse, polyline, or spline.
- 4 Enter **b** (Block).
- 5 Enter the name of the block you want to insert.
- 6 Enter **y** to align the blocks with the divided object. Enter **n** to use a rotation angle of 0 degrees.
- 7 Enter the number of segments you want.

Command line: BLOCK, DIVIDE

Extract Geometric Information from Objects

The inquiry and calculation commands can provide information about objects in your drawing and do useful calculations.

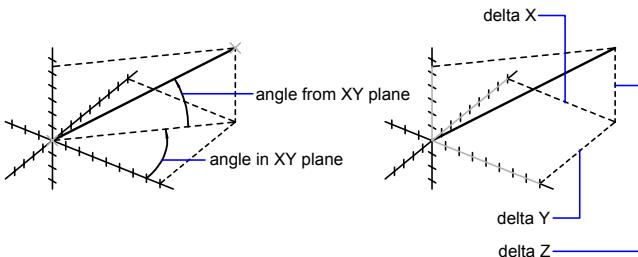
Obtain Distances, Angles, and Point Locations

You can retrieve information about the relation between two specified points: for example, the distance between them or their angle in the XY plane.

To determine the relation between two points, you can display the

- Distance between them
- Angle between the points in the XY plane
- Angle of the points from the XY plane

- Delta, or changed, X, Y, and Z distances between them



The ID command lists the X, Y, and Z coordinate values of a specified point.

See also:

[“Overview of Coordinate Entry” on page 328](#)

To calculate a distance and angle

- 1 Click Tools menu > Inquiry > Distance.
- 2 Specify the first and second points of the distance you want to calculate.
A brief report is displayed on the command line.

Inquiry toolbar



Command line: DIST

Obtain Area Information

You can obtain the area and perimeter defined by selected objects or a sequence of points.

You can calculate and display the area and perimeter of a sequence of points or of any of several types of objects.

TIP A fast way to calculate an area bounded by several objects in 2D is to use the BOUNDARY command. With BOUNDARY, you can pick a point within the area to

create a closed polyline or region. You can then use the Properties palette or the LIST command to find the area and perimeter of the polyline or region.

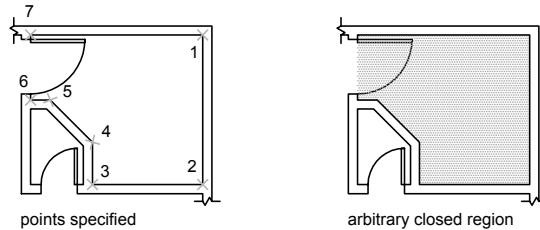
Use the AREA Command

With the AREA command, you can specify a series of points or select an object. If you need to calculate the combined area of more than one object, you can keep a running total as you add or subtract one area at a time from the selection set. You cannot use window selection or crossing selection to select objects.

Total area and perimeter are saved in the AREA and PERIMETER system variables.

Calculate a Defined Area

You can measure an arbitrary closed region defined by the points you specify. The points must lie on a plane parallel to the XY plane of the current UCS.

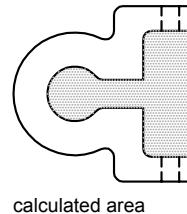
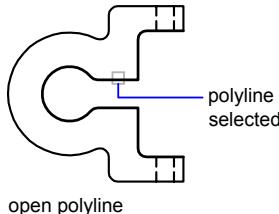
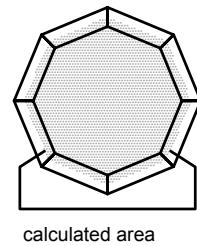
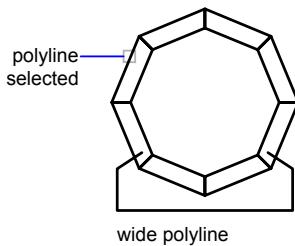
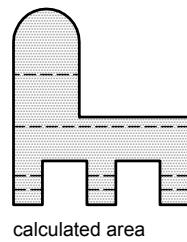
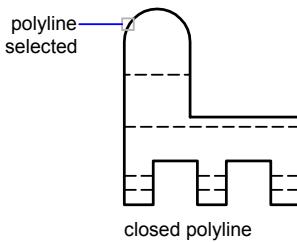


Calculate the Area, Perimeter, or Circumference of an Object

You can calculate the enclosed area and perimeter or circumference of circles, ellipses, polylines, polygons, regions, and AutoCAD 3D solids. The displayed information differs according to the type of object you select:

- **Circles.** Area and circumference are displayed.
- **Ellipses, closed polylines, polygons, planar closed spline curves, and regions.** Area and perimeter are displayed. For wide polylines, this area is defined by the center of the width.
- **Open objects such as open spline curves and open polylines.** Area and length are displayed. The area is computed as though a straight line connected the start point and endpoint.
- **AutoCAD 3D solids.** Total 3D area for the object is displayed.

Example: How Various Areas Are Calculated



Calculate Combined Areas

You can measure more than one area, either by specifying points or by selecting objects. For example, you can measure the total area of selected rooms in a floor plan.

Subtract Areas from Combined Areas

You can subtract one or more areas from a combined area that you have already calculated. In the example, the area of the floor plan is first calculated, and then a room is subtracted.

Example: Subtraction of Areas from a Calculation

In the following example, the closed polyline represents a metal plate with two large holes. You first calculate the area of the polyline and then subtract each hole. Area and perimeter or circumference of each object are displayed, with a running total after each step.

Here is the command line sequence:

Command: **area**

Specify first corner point or [Object/Add/Subtract]: **a**

Specify first corner point or [Object/Subtract]: **o**

(ADD mode) Select objects: *Select the polyline (1)*

Area = 0.34, Perimeter = 2.71

Total area = 0.34

(ADD mode) Select objects: Press ENTER

Specify first corner point or [Object/Subtract]: **s**

Specify first corner point or [Object/Add]: **o**

(SUBTRACT mode) Select objects: *Select the lower circle (2)*

Area = 0.02, Circumference = 0.46

Total area = 0.32

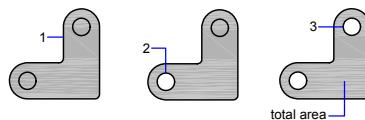
(SUBTRACT mode) Select objects: *Select the upper circle (3)*

Area = 0.02, Circumference = 0.46

Total area = 0.30

(SUBTRACT mode) Select circle or polyline: Press ENTER

Specify first corner point or [Object/Add]: Press ENTER



You can also use REGION to convert the plate and the holes to regions, subtract the holes, and then use the Properties palette or the LIST command to find the area of the plate.

TIP Use the QuickCalc calculator to convert from one system of area units to another. For more information, see Use the QuickCalc Calculator.

See also:

"Create and Combine Areas (Regions)" on page 452

"Overview of Object Properties" on page 273

To calculate an area you define

- 1 Click Tools menu ▶ Inquiry ▶ Area.
- 2 Specify points in a sequence that defines the perimeter of the area to be measured. Then press ENTER.
The first and last points are connected to form a closed area and the area and perimeter measurements are displayed using the settings specified with UNITS.

Inquiry toolbar



Command line: AREA

To calculate the area of an object

- 1 Click Tools menu ▶ Inquiry ▶ Area.
- 2 On the command line, enter **o** (Object).
- 3 Select an object.

The area and perimeter of the selected object are displayed.

Inquiry toolbar



Command line: AREA

To add areas as you calculate

- 1 Click Tools menu ▶ Inquiry ▶ Area.
- 2 Enter **a** (Add).
- 3 Use one of the following methods:
 - Specify points to define the area you want to add and press ENTER.
 - Enter **o** (Object) and select the objects you want to add.
You see the measurements of each new area and a running total of all areas.
- 4 Press ENTER twice to end the command.

Inquiry toolbar



Command line: AREA

To subtract areas from a calculation

- 1 While a combined area is displayed, enter **s** (Subtract).
- 2 Use one of the following methods:
 - Specify points to define the area you want to subtract and press ENTER.
 - Enter **o** (Object) and select the objects you want to subtract.
- The running total is updated as you define new areas.
- 3 Press ENTER to end the command.

Inquiry toolbar



Command line: AREA

Use a Calculator

You can access a calculator function as you work with the program. You can use either the QuickCalc calculator interface or the command line calculator.

Use the QuickCalc Calculator

With the QuickCalc calculator, an interface that looks and functions like a hand-held calculator, you can perform mathematical, scientific, and geometric calculations, convert units of measurement, manipulate the properties of objects, and evaluate expressions.

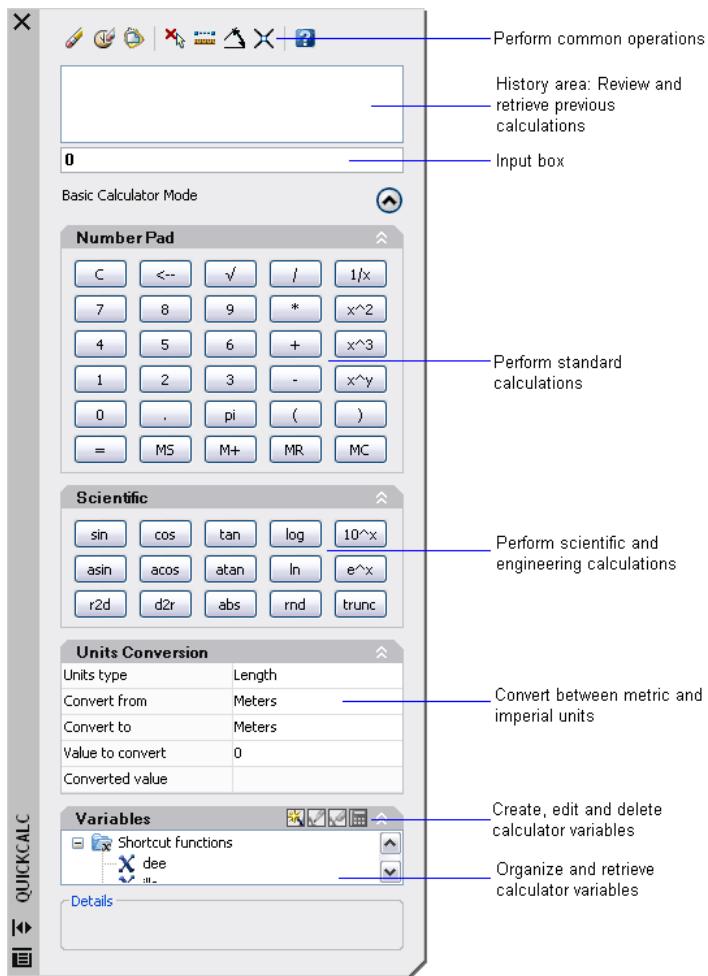
Overview of QuickCalc

The QuickCalc calculator includes basic features similar to most standard mathematical calculators. In addition, QuickCalc has features specific to AutoCAD such as geometric functions, a Units Conversion area, and a Variables area.

Unlike most calculators, QuickCalc is an expression builder. For greater flexibility, it does not immediately calculate an answer when you click a function. Instead, you compose an expression that you can easily edit and, when you are finished, you click the equal sign (=) or press ENTER. Later, you can retrieve the expression from the History area, modify it, and recalculate the results.

With QuickCalc, you can

- Perform mathematical and trigonometric calculations
- Access and review previously entered calculations for re-evaluation
- Use the calculator with the Properties palette to modify object properties
- Convert units of measurement
- Perform geometric calculations related to specific objects
- Copy and paste values and expressions to and from the Properties palette and the command line
- Perform computations on mixed numbers (fractions), and feet and inches
- Define, store, and use calculator variables
- Use geometric functions from the CAL command



Change QuickCalc Size and Appearance

Click the More/Less button on the calculator and only the Input box and History area are displayed. You can use the expand/collapse arrows to open and close areas. You can also control the size, location, and appearance of QuickCalc. See “Set Interface Options” on page 62.

To clear the Input box

Do one of the following:

- On the QuickCalc toolbar, click the Clear button.

- On the QuickCalc Number Pad, click the Clear button.
Any current value or expression in the Input box is cleared and the value is reset to 0.

To use basic math functions in QuickCalc

- 1 On the Number Pad, click a number.
- 2 Click an operator (+, -, *, /) button. Then enter the next number, and so on.
- 3 Click the equal (=) sign.

The result is displayed in the Input box. The expression and result are also displayed in the History area.

To use scientific functions in QuickCalc

- 1 On the Number Pad, enter a value.
- 2 In the Scientific area, click a function.
- 3 On the Number Pad, click the equal (=) sign.

The result is displayed in the Input box.

To get the absolute value of a number using QuickCalc

- 1 With the current value displayed, in the Scientific area click the abs button.
- 2 On the Number Pad, click the equal (=) sign.

The absolute value is displayed in the Input box.

To round a number to the nearest integer using QuickCalc

- 1 With the current value displayed, in the Scientific area click the rnd button.
- 2 On the Number Pad, click the equal (=) sign.

The number, rounded to the nearest integer, is displayed in the Input box.

To display only the integer portion of a number with QuickCalc

- 1 With the current value displayed, in the Scientific area click the trunc button.

- 2** On the Number Pad, click the equal (=) sign.

The integer portion of the number is displayed in the Input box without the decimal portion.

To store a value in the QuickCalc memory

- With the current value displayed, click MS.

The previous value is overwritten and the new value is stored in memory.

To add the current value to the value stored in the QuickCalc memory

- With the current value displayed, click M+.

The sum of the new value and the value already in memory are displayed.

To restore the value stored north QuickCalc memory

- With the current value displayed, click MR.

The displayed value or expression is appended and the value stored in memory is displayed.

To clear the value stored in the QuickCalc memory

- Click MC.

The value currently stored in memory is cleared.

Access QuickCalc and Understand Its Behavior

There are three ways to work with QuickCalc within the program:

- Directly from the Tools menu, the Standard toolbar, a shortcut menu, or the Command prompt
- Transparently during a command from a shortcut menu or the command line
- Transparently from the Properties palette

The method that you choose depends on how you are using QuickCalc.

Use QuickCalc Directly

When you work directly with QuickCalc, you can perform calculations and unit conversions just as you would with a desktop calculator. You can use the

Windows clipboard (CTRL+C, CTRL+V) to transfer the results of your calculations to other parts of the program or to external programs. Calculations that you perform directly do not affect or change anything in your drawing.

You can access QuickCalc directly in the following ways:

- On the Tools menu, click Palettes and then QuickCalc
- On the Standard toolbar, click the QuickCalc button
- In the drawing editor (with no command active), right-click, and then click QuickCalc
- On the command line, enter **quickcalc** or **qc**, and then press ENTER

Use QuickCalc Transparently from Within a Command

During a command, you can access QuickCalc transparently in the following ways:

- Right-click to display the shortcut menu. Click QuickCalc.
- Enter '**quickcalc**' or '**qc**' on the command line.

Calculations that you transfer to the command line affect the drawing. For example, if you are drawing a line with the LINE command and you pass an expression to the command line from the calculator, the next point of the line uses the results, which might be a distance or coordinate value. In QuickCalc, you click the Apply button to pass the value to the drawing based on that expression.

NOTE When using QuickCalc transparently to calculate a value for direct distance entry, Apply transfers the value to the command line. To use the value, position the crosshairs to determine the direction and then press ENTER.

Use QuickCalc Transparently from the Properties Palette

When you want to modify properties and apply evaluated expressions to objects in your drawing, you can access QuickCalc transparently from the Properties palette. Use the following method:

- Click any box in the Properties palette that contains a numeric value. Then click the QuickCalc button that appears in the box.
- After calculating a value, transfer the result to the Properties palette by clicking the Apply button.

The object or drawing property is modified and the changes are displayed in the drawing.

Calculate Mathematical Expressions in a Dialog Box

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

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

To use the QuickCalc calculator

Do *one* of the following:

- On the command line, enter **quickcalc** or **qc**.
- Right-click the drawing area to display a shortcut menu. Click QuickCalc.
- On the Tools menu, click Palettes ➤ QuickCalc.
- On the Standard toolbar, click QuickCalc.

To use the QuickCalc calculator within a command

Do *one* of the following:

- On the command line, enter '**quickcalc**' or '**qc**'.
- Right-click to display a shortcut menu. Click QuickCalc.

To use the QuickCalc calculator with the Properties palette

- 1 Open the Properties palette.
- 2 Click one or more objects.
- 3 Click a box with a numeric property.
- 4 Click the QuickCalc button in the box.

Enter, Evaluate, and Retrieve Expressions

Enter expressions in QuickCalc using standard mathematical rules of precedence; review and retrieve computations from the History area; and understand the rules for using imperial units: length, area, and volume.

QuickCalc evaluates expressions according to the following standard mathematical rules of precedence:

- Expressions in parentheses first, starting with the innermost set
- Operators in standard order: exponents first, multiplication and division second, and addition and subtraction last
- Operators of equal precedence from left to right

The Input box of the calculator is where you enter and retrieve expressions. With QuickCalc, there are two ways you can enter data in the Input box. You can either enter expressions using the QuickCalc Number Pad buttons, or you can use the computer keyboard or numeric keypad. To use the computer numeric keypad, you must have NUMLOCK on.

To evaluate an expression, click the equal (=) sign on the QuickCalc Number Pad, or press ENTER on the computer keyboard.

Understand the Syntax of Expressions

The syntax for QuickCalc expressions and command line calculator expressions is identical. For example, to perform an operation on the vector or coordinates 5,2,0, you enter [5,2,0] in the Input box.

You can use the GETVAR function to read the value of a system variable. The syntax is

`getvar(variable_name)`

For more information see CAL.

Use the History Area

The History area keeps an ongoing record of calculations, similar to the paper tape in a physical desk calculator. You can use the History area to review previous operations and pass them back to the Input box for re-evaluation with different parameters.

Rules for Displaying and Handling Units

QuickCalc adheres to the following rules:

- Results of calculations are always expressed in decimal format unless a distance is entered in feet and inches
- Angular values entered in the Input box are assumed to be degrees regardless of the settings in the Drawing Units dialog box. To specify radians, grads, and degrees, append an **r**, **g**, or **d** after the angle value.

- Results of angular calculations are always expressed in degrees with full AutoCAD precision.

When the drawing units are set to architectural units, the calculator displays the results of calculations of imperial units in the architectural format and rounds to the display precision (LUPREC) specified in the drawing. The results for all other calculations display in decimal format with full precision.

You can separate feet, inches, and fractional inches with a dash, a space, or nothing. You can use any of the following syntax cases to enter valid feet-inch formatted values:

- 5' or 60"
- 5'-9" or 5' 9" or 5'9"
- 5'-1/2" or 5' 1/2" or 5'1/2"
- 5'-9-1/2" or 5' 9-1/2" or 5'9-1/2"
- 5'-9 1/2" or 5' 9 1/2" or 5'9 1/2"

To designate inches for linear calculations, entering double quotes ("") is optional. For example, instead of entering 5'9-1/2", you could enter 5'9-1/2.

WARNING With imperial units, QuickCalc interprets a minus or a dash (-) as a unit separator rather than a subtraction operation. To specify subtraction, include at least one space before or after the minus sign. For example, to subtract 9" from 5', enter **5' -9"** rather than **5'-9"**.

You can use QuickCalc to calculate square feet and cubic feet. To enter square or cubic feet, you must enter units using these abbreviations:

- **sq. ft. or sq ft**
- **cu. ft. or cu ft**

To copy and paste a value from the QuickCalc Input box

- Right-click the value in the Input box. Click Copy.
The current entry is copied to the clipboard.
- Click in the new location and then right-click. Click Paste.
The value is copied to the new location.

To paste a value from the QuickCalc Input box to the command line

- 1 On the QuickCalc toolbar, click the Paste Value to Command Line button.

The value in the Input box is pasted into the command line.

To clear the History area

- On the QuickCalc toolbar, click the Clear History button.

Shortcut menu: Right-click the History area. Click Clear History.

To reuse a stored value or expression in the QuickCalc History area

- 1 Click the C button to clear the Input box if needed.
- 2 In the History area, double-click a value or expression. The value or expression is displayed in the Input box.

NOTE The cursor must be on the value or expression to select it.

To change the font color of values or expressions in the QuickCalc History area

- 1 Right-click in the History area. Click either Value Font Color or Expression Font Color.
 - 2 In the Color dialog box, click a Basic Color or click Define Custom Colors. With Define Custom Colors, you can select a custom color and add it to Custom Colors.
 - 3 Click OK.
- The colors you selected for values and expressions in the History area are displayed.
- Shortcut menu:** Right-click Expression Font Color or Value Font Color. Click Basic Color or Define Custom Colors.

To copy and paste an expression from the QuickCalc History area

- Right-click the expression in the History area. Click Copy.
The current entry is copied to the clipboard.
- Right-click in the new location. Click Paste.
The expression is copied to the new location.

To append a value or expression from the QuickCalc History area to the Input box

- Double-click the value or expression in the History area.

The value or expression is appended to the Input box.

NOTE The cursor must be on the value or expression to select it.

Shortcut menu: Right-click Append Value to Input Area or Append Expression to Input Area.

To modify a property in the Properties palette with QuickCalc

- 1 Select an object.

NOTE If QuickCalc is displayed on your desktop when you work with the Properties palette, it is temporarily hidden while you use the modal calculator from within the Properties palette.

- 2 In the Geometry section of the Properties palette, click on the value of a property.

A small calculator icon is displayed to the right of the value.

NOTE Only properties that are displayed with a white background can be changed.

- 3 Click the calculator icon.

QuickCalc opens and displays the current value of the object in the Input box.

- 4 Perform a calculation on the displayed value and click the equal sign (=) button.

The new value is displayed in the Input box.

- 5 Click Apply.

NOTE The Apply button is only available for editable number-based properties.

The calculator closes and the new value is displayed in the Properties palette. The object is modified in the drawing.

To obtain X, Y, Z coordinate values for a point using QuickCalc

NOTE The QuickCalc toolbar Get Coordinates button uses the `cur` function.

- 1 On the QuickCalc toolbar, click the Get Coordinates button.
QuickCalc temporarily closes and you are prompted to specify a point.
- 2 In the drawing, click a point.
QuickCalc opens and displays the coordinate values of the point in the Input box.

To measure the distance between two points using QuickCalc

NOTE The QuickCalc toolbar Distance Between Two Points button uses the `dist(p1,p2)` function.

- 1 On the QuickCalc toolbar, click the Distance Between Two Points button.
QuickCalc temporarily closes and you are prompted to specify two points.
- 2 In the drawing, click the first point and then the second point.
QuickCalc opens and displays the value of the distance between the two points in the Input box.

To get the angle of a line defined by two points using QuickCalc

NOTE The QuickCalc toolbar Angle of Line Defined by Two Points button uses the `ang(p1,p2)` function.

- 1 On the QuickCalc toolbar, click the Angle of Line Defined by Two Points button.
QuickCalc temporarily closes and you are prompted at the command line to specify two points.
- 2 Enter the coordinate values for the first point and then the second point.
QuickCalc opens and appends the value of the angle between the two points to the end of any value or expression already present in the Input box.

To get the intersection of a line defined by four points using QuickCalc

NOTE The QuickCalc Intersection of Two Lines Defined by Four Points button uses the $\text{ill}(p1,p2,p3,p4)$ function.

- 1 On the QuickCalc toolbar, click the Intersection of Two Lines Defined by Four Points button.
QuickCalc temporarily closes and you are prompted at the command line to specify four points.
- 2 Enter the coordinate values for the first point of line one, then the second point of line one. Next, enter the coordinate values for the first point of line two, then the second point of line two.
QuickCalc opens and appends the value of the evaluated expression to the end of any value or expression already present in the Input box.

Convert Units of Measurement

In the Units Conversion area of QuickCalc, you can obtain equivalent values for different units of measurement.

Unit conversions are available for length, area, volume, and angular values. Based on which unit type you select, you can then select a list of units to convert from and a list of units to convert to.



The Value to Convert box automatically displays the value from the Input box. You can also enter a different value. The results of the units conversion displays in the Converted Value box. You can paste this result to the Input box by clicking the QuickCalc icon in the Converted Value box.

NOTE In the Value to Convert box, enter decimal values without units.

To convert units of measurement with QuickCalc

- 1 In the Units Conversion area, select a unit category on the Units Type list.
- 2 In the Convert From list, select the type of unit you are converting from.
- 3 In the Convert To list, select the type of unit you are converting to.
- 4 In the Value to Convert box, enter the value you want to convert. Press ENTER.

The converted value is displayed in the Converted Value box.

To copy a unit conversion result to the Input box of QuickCalc

- 1 On the Units Conversion title bar, click the Return Conversion Value to Input Area button.

The converted value is displayed in the Input box.

To convert radians to degrees using QuickCalc

- 1 On the Number Pad, enter a value in radians.
- 2 In the Scientific area, click the r2d button.
- 3 On the Number Pad, click the equal (=) sign.

The conversion is displayed in the Input box.

To convert degrees to radians using QuickCalc

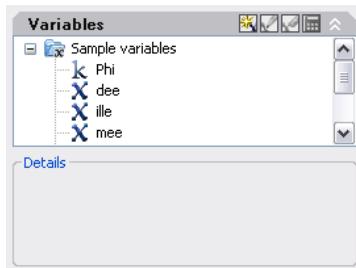
- 1 On the Number Pad, enter a value in degrees.
- 2 In the Scientific area, click the d2r button.
- 3 On the Number Pad, click the equal (=) sign.

The conversion is displayed in the Input box.

Create and Use Calculator Variables

You can use the Variables area to define, store, and retrieve *calculator variables*. Calculator variables can either be constants (coordinates/vectors, real numbers, and integers) or functions. In the Variables area, you can

- Click a calculator variable to display information such as value, type, and description in the Details box at the bottom of the Variables area.
- Double-click a calculator variable to load it into the QuickCalc Input box.



Additional operations are available on the shortcut menus in the Variables area.

Create New Calculator Variables

You can create new calculator variables using the shortcut menus in the Variables area. When defining new calculator variables in the Variable Definition dialog box, the following rules apply:

- **Constants.** Any expression entered in the Value or Expression text entry box is evaluated before the calculator variable is stored. Calculator variables that are defined as constants are available “globally.” You can access and use global constants in different drawings and sessions.
- **Functions.** Any expression entered in the Value or Expression text entry box is stored as text. Functions are evaluated when used in the QuickCalc Input box.

Create Global Constants

You can use one of the following methods to create global constants:

- Enter an expression in the Input box using the format **\$ variable_name = value**. For example, to define the golden ratio to 8 decimal places as a global constant called Phi, enter **\$Phi=1.61803399** in the Input box.
- Click the New Variable button in the Variables area title bar. In the Variables Definition dialog box, click Constant and fill in the other boxes.
- Right-click the Variables area. Click New Variable.

Access Global Constants

You can access global constants and pass them to the Input box of the QuickCalc as follows:

- Double-click a variable in the Variables area of QuickCalc.
- Click a variable from the list of calculator variables, and click the Return Variable to Input Area button.
- Enter a dollar sign (\$) followed by the variable name, and press ENTER.

To use a global constant in a text or numeric entry box in a window or dialog box, use the syntax: **=\$ variable_name** followed by pressing the END key. For example, to use the previously mentioned global variable, Phi, enter **=\$Phi** and press the END key.

NOTE In QuickCalc, only constants can be directly referred to by their global variable names in the text or numeric entry boxes in windows and dialog boxes.

Use Shortcut Functions

Several sample calculator variables have been predefined and stored in the Shortcut Functions category. These are geometric expressions that combine CAL functions with the Endpoint Snap mode. The following table describes the predefined variables that are available in the Variables area of the calculator.

Variable	Shortcut For	Description
dee	dist(end,end)	Distance between two endpoints
ille	ill(end,end,end)	Intersection of two lines defined by four endpoints
mee	(end+end)/2	Midpoint between two endpoints
nee	nor(end,end)	Unit vector in the XY plane and normal to two endpoints
rad	rad	Radius of a selected circle, arc, or polyline arc
vee	vee(end,end)	Vector from two endpoints
vee1	vec1(end,end)	Unit vector from two endpoints

You can easily modify these calculator variables or create your own. For more information, see the CAL command.

Organize Variables into Categories

You can organize calculator variables in the Variables area under several categories. This results in a one-level tree structure. The Shortcut Functions category has already been created and contains several functions.

Use the shortcut menu in the Variables area to create, rename, or delete variable categories.

To use a predefined variable in a QuickCalc expression

- 1 In the Variables area, click the variable you want to use.
- 2 On the QuickCalc Variables title bar, click the Return Variable to Input Area button.

The variable is displayed in the Input box as part of your expression.

To create a new variable in QuickCalc

- 1 On the QuickCalc Variables title bar, click the New Variable button.
- 2 In the Variable Definition dialog box, under Variable Type, select Constant or Function.
- 3 In the Variable Definition dialog box, under Variable Properties Name, enter a name for the variable. Variable names cannot contain spaces or special characters.

NOTE When referencing this variable from the Input box, the variable name must begin with the dollar sign (\$) to differentiate it from local LISP variables.

- 4 Under Variable Properties Group With, click New.
- 5 In the Category Definition dialog box, under Category Properties Name, enter a name for the new category.
- 6 Under Description, enter a description for the new category. Click OK.
- 7 In the Variable Definition dialog box, under Value or Expression, enter a value or expression for the new variable.
- 8 Under Description, enter a description of the new variable. Click OK.
The new variable is now displayed in the Variables area.

To edit a variable in QuickCalc

- 1 In the Variables area, click the variable you want to edit.
- 2 On the QuickCalc Variables title bar, click the Edit Variable button.
- 3 In the Variable Definition dialog box, make the edits to the variable. Click OK.

To delete a variable in QuickCalc

- 1 In the Variables area, click the variable you want to delete.
- 2 On the QuickCalc Variables title bar, click the Delete button.

To create a new global constant in the Input box of QuickCalc

- 1 In the Input box of QuickCalc, enter the following syntax: **\$variable_name = value**.

For example, you could center **\$Phi=1.618**

NOTE Global variables are not case-sensitive.

QuickCalc adds the global constant to the list of variables in the Variables area.

To access a global constant from dialog box or window

- In any text or numeric entry box, enter an expression using the syntax: =\$*variable_name* followed by pressing the END key.

To evaluate mathematical expressions in a dialog box

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

- In a dialog box, in an area where you can enter a numeric value, enter a mathematical expression in the following format: =*expression* (for example, =5+3).
- Click END on the keyboard.
The expression is evaluated and its value is displayed.

Command line: CALCINPUT

Use the Command Line Calculator

By entering an expression in the command line calculator, you can quickly solve a mathematical problem or locate points in your drawing.

The CAL command runs the 3D calculator utility to evaluate vector expressions (combining points, vectors, and numbers) and real and integer expressions. The calculator performs standard mathematical functions. It also contains a set of specialized functions for calculations involving points, vectors, and AutoCAD geometry. With the CAL command, you can

- Calculate a vector from two points, the length of a vector, a normal vector (perpendicular to the XY plane), or a point on a line
- Calculate a distance, radius, or angle
- Specify a point with the pointing device
- Specify the last-specified point or intersection
- Use object snaps as variables in an expression
- Convert points between a UCS and the WCS
- Filter the X, Y, and Z components of a vector

- Rotate a point around an axis

Evaluating Expressions

CAL evaluates expressions according to standard mathematical rules of precedence.

Mathematical operators in order of precedence

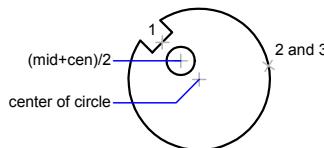
Operator	Operation
()	Groups expressions
^	Indicates numeric exponent
* /	Multiplies and divides numbers
+ -	Adds and subtracts numbers

Calculating Points

You can use CAL whenever you need to calculate a point or a number within a command.

For example, you enter **(mid+cen)/2** to specify a point halfway between the midpoint of a line and the center of a circle.

The following example uses CAL as a construction tool. It locates a center point for a new circle, and then calculates one fifth of the radius of an existing circle.



Here is the command line sequence:

Command: **circle**

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

>> Expression: **(mid+cen)/2**

>> Select entity for MID snap: *Select the notch line (1)*

>> Select entity for CEN snap: *Select the large circle (2)*

Diameter/<Radius>: '**cal**

>> Expression: **1/5*rad**

>> Select circle, arc or polyline segment for RAD function: *Select the large circle (3)*

Calculate Mathematical Expressions in a Dialog Box

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

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

No procedures for this topic.

Draw Geometric Objects

I 9

You can create a range of objects, from simple lines and circles to spline curves, and ellipses. In general, you draw objects by specifying points with the pointing device or by entering coordinate values on the command line.

In this chapter

- Draw Linear Objects
- Draw Curved Objects
- Draw Construction and Reference Geometry
- Create and Combine Areas (Regions)
- Create Revision Clouds

Draw Linear Objects

A line, the most basic object, can be one segment or a series of connected segments.

Draw Lines

In a simple line with connected segments, each segment is a separate line object.

With LINE, you can create a series of contiguous line segments.

Each single line segment can be edited separately from the other line segments in a series. You can close a sequence of line segments so that the first and last segments are joined.

You can assign properties to lines including color, linetype, and linewidth. For more information about properties, see “Control the Properties of Objects” on page 271.

You specify the locations that define the endpoints of each line with precision. You can

- Enter the coordinate values for an endpoint, using either absolute or relative coordinates
- Specify an object snap relative to an existing object. For example, you can specify the center of a circle as one endpoint of the line
- Turn grid snap on and snap to a location

There are other methods for creating precise lines. A highly efficient technique is to offset a line from an existing line, and then trim or extend it to the desired length.

Use polyline objects instead of line objects if you want the segments to be connected as a single object.

See also:

- “Use Coordinates and Coordinate Systems (UCS)” on page 328
- “Use Object Snaps” on page 364
- “Adjust Grid and Grid Snap” on page 371
- “Draw Polylines” on page 419
- “Offset an Object” on page 627
- “Break and Join Objects” on page 652

To draw lines

1 Click Draw menu ► Line.

2 Specify the start point.

You can use the pointing device or enter coordinate values on the command line.

3 Complete the first line segment by specifying the endpoint.

To undo the previous line segment during the LINE command, enter **u** or click Undo on the toolbar.

4 Specify the endpoints of any additional line segments.

5 Press ENTER to end or **c** to close a series of line segments.

To start a new line at the endpoint of the last line drawn, start the LINE command again and press ENTER at the Specify Start Point prompt.

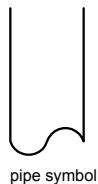
Draw toolbar



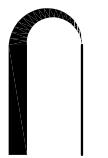
Command line: LINE

Draw Polylines

A polyline is a connected sequence of line segments created as a single object. You can create straight line segments, arc segments, or a combination of the two.



pipe symbol



differing widths

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.

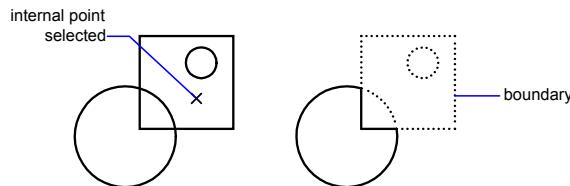
Taper

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 define the boundary.



See also:

- “Draw Rectangles and Polygons” on page 423
- “Modify Complex Objects” on page 664
- “Break and Join Objects” on page 652
- “Control Lineweights” on page 317

To draw a polyline with straight segments

- 1 Click Draw menu ▶ Polyline.
- 2 Specify the first point of the polyline.
- 3 Specify the endpoint of the first polyline segment.
- 4 Continue specifying segment endpoints as needed.
- 5 Press ENTER to end, or enter **c** to close the polyline.

To start a new polyline at the endpoint of the last polyline drawn, start the PLINE command again and press ENTER at the Specify Start Point prompt.

Draw toolbar



Command line: PLINE

To draw a line and arc combination polyline

- 1 Click Draw menu ► Polyline.
- 2 Specify the start point of the polyline segment.
- 3 Specify the endpoint of the polyline segment.
 - Switch to Arc mode by entering **a** (Arc) on the command line.
 - Return to Line mode by entering **L** (Line).
- 4 Specify additional polyline segments as needed.
- 5 Press ENTER to end, or enter **c** to close the polyline.

Draw toolbar



Command line: PLINE

To create a wide polyline

- 1 Click Draw menu ► Polyline.
- 2 Specify the start point of the line segment.
- 3 Enter **w** (Width).
- 4 Enter the starting width of the line segment.
- 5 Specify the ending width of the line segment using one of the following methods:
 - To create a line segment of equal width, press ENTER.
 - To create a tapering line segment, enter a different width.
- 6 Specify the endpoint of the polyline segment.
- 7 Continue specifying segment endpoints as needed.
- 8 Press ENTER to end, or enter **c** to close the polyline.

Draw toolbar



Command line: PLINE

To create a boundary polyline

- 1 Click Draw menu ► Boundary.
- 2 In the Boundary Creation dialog box, in the Object Type list, select Polyline.
- 3 Under Boundary Set, do one of the following:
 - To create a boundary set from all objects visible in the current viewport, select Current Viewport. Avoid this option for large, complex drawings.
 - To specify which objects to include in the boundary set, click New. Select the objects that you want to use to create the boundary. Using this option automatically selects the Existing Set option.
- 4 Click Pick Points.
- 5 Specify points within each area that you want to form a boundary polyline.

This area must be totally enclosed; that is, there can be no gaps between enclosing objects. You can select more than one area. Click Island Detection if you want internal closed areas to be included in the boundary set.
- 6 Press ENTER to create the boundary polyline and end the command.

The command creates a polyline in the shape of the boundary. Because this polyline overlaps the objects used to create it, it may not be visible. However, you can move, copy, or modify it just as you can any other polyline.

Command line: BOUNDARY

Draw Rectangles and Polygons

You can create rectangles and regular polygons quickly. Creating polygons is a simple way to draw equilateral triangles, squares, pentagons, hexagons, and so on.

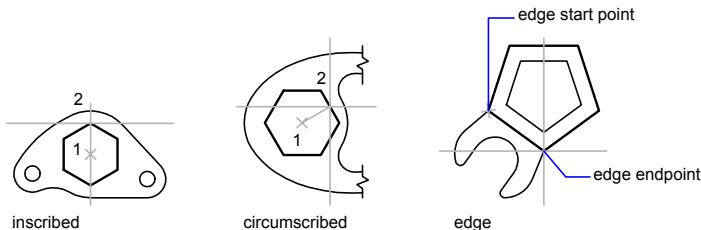
Two commands, RECTANG and POLYGON, provide an efficient method to create rectangles and regular polygons such as equilateral triangles, squares, pentagons, hexagons, and so on. If necessary, you can use EXPLODE to convert the resulting polyline object into lines.

Draw Rectangles

Use **RECTANG** to create closed polylines in a rectangular shape. You can specify the length, width, area, and rotation parameters. You can also control the type of corners on the rectangle—fillet, chamfer, or square.

Draw Regular Polygons

Use **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.

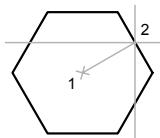


See also:

["Draw Polylines" on page 419](#)

To draw a circumscribed polygon

- 1 Click Draw menu ▶ Polygon.
- 2 On the command line, enter the number of sides.
- 3 Specify the center of the polygon (1).
- 4 Enter **c** to specify a polygon circumscribed about a circle.
- 5 Enter the radius length (2).



Draw toolbar



Command line: POLYGON

To draw a polygon by specifying one edge

- 1 Click Draw menu ► Polygon.
- 2 On the command line, enter the number of sides.
- 3 Enter **e** (Edge).
- 4 Specify the start point for one polygon segment.
- 5 Specify the endpoint of the polygon segment.

Draw toolbar



Command line: POLYGON

To draw an inscribed polygon

- 1 Click Draw menu ► Polygon.
- 2 On the command line, enter the number of sides.
- 3 Specify the center of the polygon.
- 4 Enter **i** to specify a polygon inscribed within a circle of specified points.
- 5 Enter the radius length.

Draw toolbar



Command line: POLYGON

To draw a rectangle

- 1 Click Draw menu ► Rectangle.
- 2 Specify the first corner of the rectangle.
- 3 Specify the other corner of the rectangle.

Draw toolbar



Command line: RECTANG

Draw Multiline Objects

Multilines are composed of 1 to 16 parallel lines, called elements.

When you draw a multiline, you can use the STANDARD style, which has two elements, or specify a style that you created previously. You can also change the justification and scale of the multiline before you draw it.

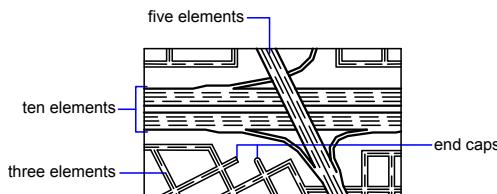
Multiline justification determines which side of the cursor that the multiline is drawn, or whether it is centered on the cursor.

Multiline scale controls the overall width of the multiline using the current units. Multiline scale does not affect linetype scale. If you change the multiline scale, you might need to make equivalent changes to the linetype scale to prevent dots or dashes from being disproportionately sized.

Create Multiline Styles

You can create named styles for multilines to control the number of elements and the properties of each element. The properties of multilines include

- The total number of elements and position of each element
- The offset distance for each element from the middle of the multiline
- The color and linetype of each element
- The visibility of the lines, called *joints*, that appear at each vertex
- The type of end caps that are used
- The background fill color of the multiline



You can add up to 16 elements to a multiline style. Elements with a positive offset appear on one side of the middle of the multiline; elements with a negative offset appear on the other side of the middle of the multiline.

See also:

["Modify Multilines" on page 673](#)

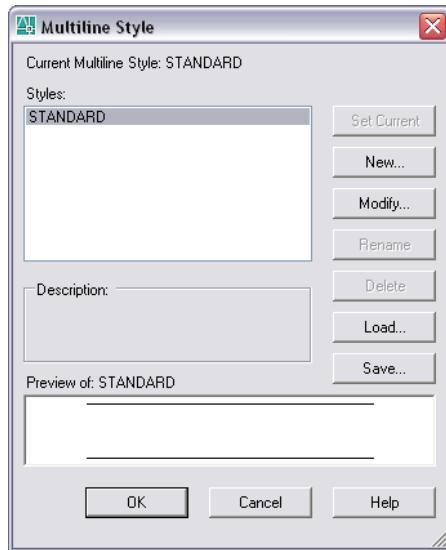
To draw a multiline

- 1 Click Draw menu ▶ Multiline.
- 2 At the Command prompt, enter **st** to select a style.
- 3 To list available styles, enter the style name or enter **?**.
- 4 To justify the multiline, enter **j** and select top, zero, or bottom justification.
- 5 To change the scale of the multiline, enter **s** and enter a new scale.
Now draw the multiline.
- 6 Specify the starting point.
- 7 Specify a second point.
- 8 Specify additional points, or press ENTER. If you specify three or more points, you can enter **c** to close the multiline.

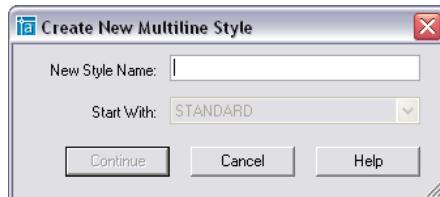
Command line: MLINE

To create a multiline style

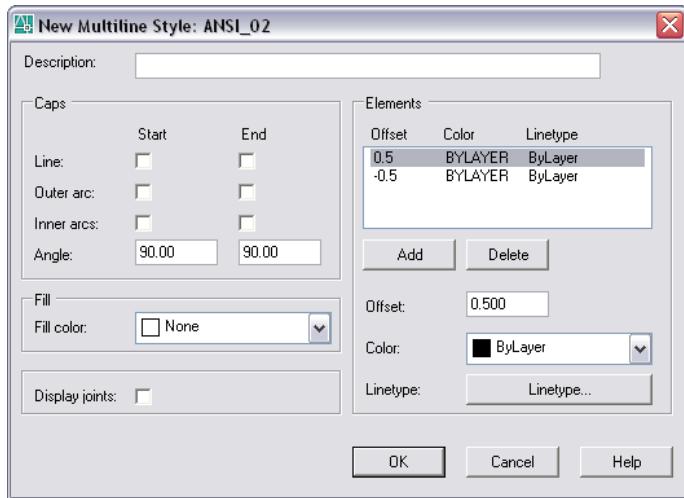
- 1 Click Format menu ▶ Multiline Style.



- 2 In the Multiline Style dialog box, click New.
- 3 In the Create New Multiline Style dialog box, enter a name for the multiline style and select a multiline style from which to start. Click Continue.



- 4 In the New Multiline Style dialog box, select the parameters for the multiline style. You can also enter a description.
Descriptions are optional and can be up to 255 characters, including spaces.



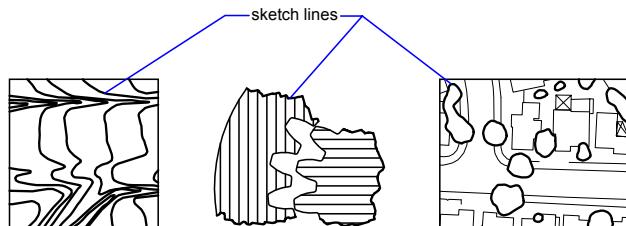
- 5 Click OK.
- 6 In the Multiline Style dialog box, click Save to save the multiline style to a file (the default is *acad.mln*). You can save multiline styles to the same file.

If you create more than one multiline style, save the current style before creating a new one or you lose the changes to the first style.

Draw Freehand Sketches

Sketching is useful for creating irregular boundaries or for tracing with a digitizer.

You can use the SKETCH command to draw freehand sketches. Sketching is useful for creating irregular boundaries or for tracing with a digitizer.



Create Sketches

To sketch, use the pointing device like a pen, clicking to put the “pen” down on the screen to draw and clicking again to lift it up and stop drawing.

Freehand sketches comprise many line segments. Each line segment can be a separate object or a polyline. You set the minimum length or increment of the segments. Small line segments allow for greater accuracy, but they can greatly increase the drawing file size. For this reason, use this tool sparingly.

Before sketching, check the CELTYPE system variable to make sure the current linetype is BYLAYER. If you use a linetype with dots or dashes and set the sketch line segment shorter than the spaces or dashes, you won't see the spaces or dashes.

Erase Freehand Lines

You erase freehand lines by using the Erase option of the SKETCH command. In Erase mode, wherever the cursor intersects the freehand line, everything from the intersection to the end of the line is erased.

Once you record freehand lines, you can't edit them or erase them with the Erase option of SKETCH. Use the ERASE command after you finish sketching.

Sketch in Tablet Mode

You use Tablet mode with a digitizer. Sketching in Tablet mode is useful for such things as tracing map outlines from paper directly into a drawing. You can't turn off Tablet mode while sketching.

When Tablet mode is on, you can configure the program to map the paper drawing's coordinate system directly into the world coordinate system. Thus, there is a direct correlation between the coordinates where screen crosshairs appear, the coordinates on the tablet, and the coordinates in the original paper drawing. After configuring the program to match the coordinates of the paper drawing, you may find that the area shown on the screen is not the area you need. To avoid this problem, use ZOOM to display the entire work area before you start to sketch.

With some digitizers you can't select the menus while Tablet mode is on. See your digitizer documentation for details.

Maintain Sketching Accuracy

To ensure accuracy on a slow computer, set the record increment value to a negative value. SKETCH uses this value as if it was positive but tests every point received from the pointer against twice the record increment. If the point is more than two record increments away, your computer beeps as a warning that you should slow down to avoid losing accuracy. For example, if the record increment is -1, you should move the cursor in increments of no more than 2. Using this method does not slow down the tracing speed.

To sketch and record freehand lines

- 1 At the Command prompt, enter **sketch**.
- 2 At the Record Increment prompt, enter the minimum line segment length.
- 3 Click the start point to put the “pen” down.

When you move the pointing device, temporary freehand line segments of the length you specified are drawn. SKETCH doesn't accept coordinate input. During the command, freehand lines are displayed in a different color.

- 4 Click the endpoint to lift the “pen” up so that you can move the cursor around the screen without drawing. Click a new start point to resume drawing from the new cursor position.
- 5 Enter **r** at any time to record (save) the line you're drawing in the database and those already drawn.
If the pen is down, you can continue drawing after recording. If the pen is up, click to resume drawing. The freehand line starts from wherever the cursor is when you click.
- 6 Press **ENTER** to complete the sketch and record all unrecorded lines.

To erase freehand lines

- 1 While running the SKETCH command, with the pen up or down, enter **e** (Erase).

If the pen was down, it moves up.

- 2 Move the cursor to the end of the line you drew last and then move it back as far along the line as you want to erase.
- 3 To end the erasure and return to the SKETCH Command prompt, enter **p**. To undo the erasure, enter **e**.

If you want to change the current viewport while sketching, make sure the pen is up, all lines entered so far have been recorded, and Tablet mode is off.

Draw Curved Objects

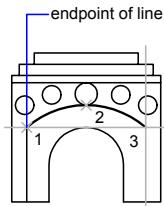
Curved objects are arcs, circles, polyline arcs, donuts, ellipses, and splines.

Draw Arcs

You can create arcs in several ways. With the exception of the first method, arcs are drawn counterclockwise from the start point to the endpoint.

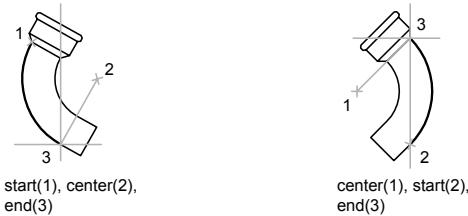
Draw Arcs by Specifying Three Points

You can draw an arc by specifying three points. In the following example, the start point of the arc snaps to the endpoint of a line. The second point of the arc snaps to the middle circle in the illustration.



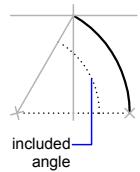
Draw Arcs by Specifying Start, Center, End

When you know the start point, center point, and endpoint, you can draw an arc by specifying either the start point or the center point first. The center point is the center of a circle that the arc is part of.

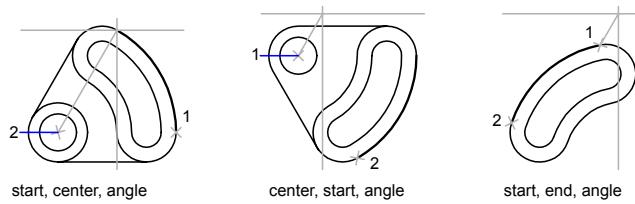


Draw Arcs by Specifying Start, Center, Angle

When you have a start point and a center point you can snap to and you know the included angle, use the Start, Center, Angle or the Center, Start, Angle option.

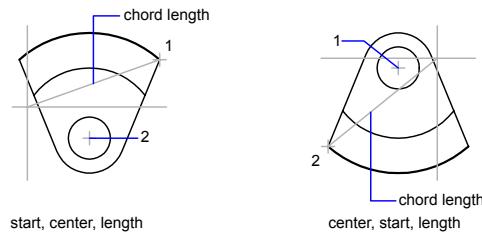


The included angle determines the endpoint of the arc. Use the Start, End, Angle method when you know both endpoints but cannot snap to a center point.



Draw Arcs by Specifying Start, Center, Length

When you have a start point and a center point you can snap to; when you know the chord length, use the Start, Center, Length or the Center, Start, Length option.

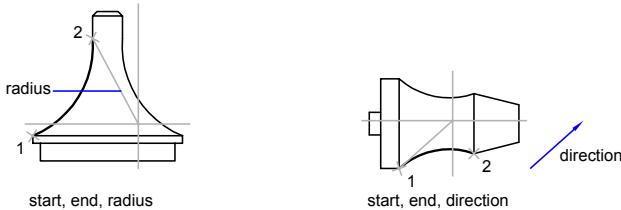


The length of the chord of the arc determines the included angle.

Draw Arcs by Specifying Start, End, Direction/Radius

When you have a start point and an endpoint, use the Start, End, Direction or the Start, End, Radius option.

The illustration on the left shows an arc drawn by specifying a start point, endpoint, and radius. You can specify the radius by entering a length or by moving the pointing device clockwise or counterclockwise and clicking to specify a distance.



The illustration on the right shows an arc drawn with the pointing device by specifying a start point and an endpoint and a direction. Moving the cursor up from the start point and endpoint draws the arc concave to the object, as shown here. Moving the cursor down draws the arc convex to the object.

Draw Contiguous Arcs and Lines

Immediately after you complete an arc, you can start a line tangent to the arc at an endpoint by starting the LINE command and pressing ENTER at the Specify First Point prompt. You need to specify only the line length.



Conversely, after you complete a line, you can start an arc tangent to the line at an endpoint by starting the ARC command and pressing ENTER at the Specify Start Point prompt. You need to specify only the endpoint of the arc.

You can connect sequentially drawn arcs in the same way. To create connected arcs using a menu, click Arc from the Draw menu, and then click Continue. In both cases, the resulting object is tangent to the previous one. You can use the shortcut menu to repeat the Continue option.

See also:

- “Draw Polylines” on page 419
- “Break and Join Objects” on page 652

To draw an arc by specifying three points

- 1 Click Draw menu ► Arc ► 3 Points.
- 2 Specify the start point.
- 3 Specify a point on the arc.
- 4 Specify the endpoint.

Draw toolbar



Command line: ARC

To draw an arc using a start point, a center point, and an endpoint

- 1 Click Draw menu ► Arc ► Start, Center, End.
- 2 Specify a start point.
- 3 Specify the center point.
- 4 Specify the endpoint.

Draw toolbar



Command line: ARC

To continue an arc with a tangential line

- 1 Complete the arc.
- 2 Click Draw menu ► Line.
- 3 Press ENTER at the first prompt.
- 4 Enter the length of the line and press ENTER.

Draw toolbar



Command line: LINE

To continue an arc with a tangential arc

- 1 Complete the arc.
- 2 Click Draw menu ► Arc ► Continue.
- 3 Specify the second endpoint of the tangent arc.

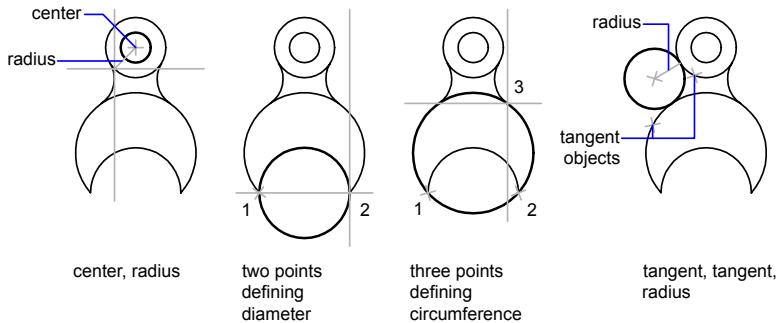
Draw toolbar



Command line: ARC

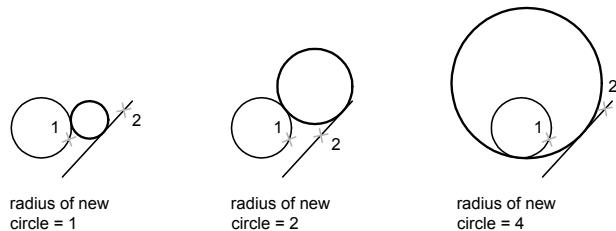
Draw Circles

You can create circles in several ways. The default method is to specify the center and the radius. Three other ways to draw a circle are shown in the illustration.



Draw a Circle Tangent to Other Objects

The tangent point is a point where an object touches another object without intersecting it. To create a circle that is tangent to other objects, select the objects and then specify the radius of the circle. In the illustrations below, the bold circle is the one being drawn, and points 1 and 2 select the objects to which it is tangent.



To create a circle tangent at three points, set running object snaps (OSNAP) to Tangent and use the three-point method to create the circle.

See also:

["Use Object Snaps" on page 364](#)

["Draw Isometric Circles"](#)

To draw a circle by specifying a center point and radius or diameter

- 1 Click Draw menu ▶ Circle ▶ Center, Radius (or Center, Diameter).
- 2 Specify the center point.
- 3 Specify the radius or diameter.

Draw toolbar



Command line: CIRCLE

To create a circle tangent to two objects

- 1 Click Draw menu ▶ Circle ▶ Tan, Tan, Radius.
The command starts Tangent object snap mode.
- 2 Select the first object to draw the circle tangent to.
- 3 Select the second object to draw the circle tangent to.
- 4 Specify the radius of the circle.

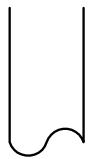
Draw toolbar



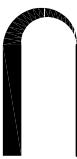
Command line: CIRCLE

Draw Polyline Arcs

A polyline is a connected sequence of line segments created as a single object. You can create straight line segments, arc segments, or a combination of the two.



pipe symbol



differing widths

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.

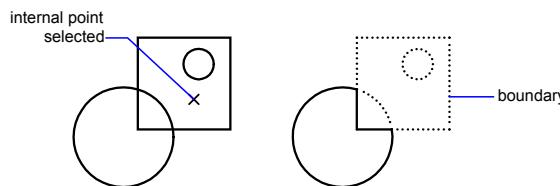
Taper

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 669
- “[Modify or Join Polylines](#)” on page 665
- “[Break and Join Objects](#)” on page 652
- “[Control Lineweights](#)” on page 317

To draw a polyline with straight segments

- 1 Click Draw menu ► Polyline.
- 2 Specify the first point of the polyline.
- 3 Specify the endpoint of the first polyline segment.
- 4 Continue specifying segment endpoints as needed.
- 5 Press ENTER to end, or enter **c** to close the polyline.
To start a new polyline at the endpoint of the last polyline drawn, start the PLINE command again and press ENTER at the Specify Start Point prompt.

Draw toolbar



Command line: PLINE

To draw a line and arc combination polyline

- 1 Click Draw menu ► Polyline.
- 2 Specify the start point of the polyline segment.
- 3 Specify the endpoint of the polyline segment.
 - Switch to Arc mode by entering **a** (Arc) on the command line.
 - Return to Line mode by entering **L** (Line).
- 4 Specify additional polyline segments as needed.
- 5 Press ENTER to end, or enter **c** to close the polyline.

Draw toolbar



Command line: PLINE

To create a wide polyline

- 1 Click Draw menu ► Polyline.
- 2 Specify the start point of the line segment.
- 3 Enter **w** (Width).
- 4 Enter the starting width of the line segment.
- 5 Specify the ending width of the line segment using one of the following methods:
 - To create a line segment of equal width, press ENTER.
 - To create a tapering line segment, enter a different width.
- 6 Specify the endpoint of the polyline segment.
- 7 Continue specifying segment endpoints as needed.
- 8 Press ENTER to end, or enter **c** to close the polyline.

Draw toolbar



Command line: PLINE

To create a boundary polyline

- 1 Click Draw menu ► Boundary.
- 2 In the Boundary Creation dialog box, in the Object Type list, select Polyline.
- 3 Under Boundary Set, do one of the following:
 - To create a boundary set from all objects visible in the current viewport, select Current Viewport in the list. Avoid this option for large, complex drawings.
 - To specify which objects to include in the boundary set, click New. Select the objects that you want to use to create the boundary. Choosing this option automatically selects the Existing Set option.
- 4 Click Pick Points.
- 5 Specify points within each area that you want to form a boundary polyline.

This area must be totally enclosed; that is, there can be no gaps between enclosing objects. You can select more than one area.

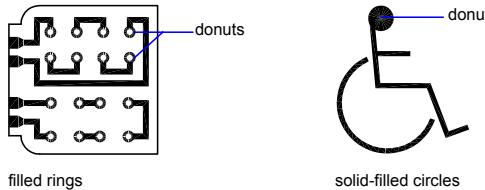
- 6 Press ENTER to create the boundary polyline and end the command.
The command creates a polyline in the shape of the boundary. Because this polyline overlaps the objects used to create it, it may not be visible. However, you can move, copy, or modify it just as you can any other polyline.

Command line: BOUNDARY

Draw Donuts

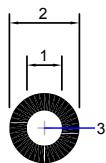
Donuts are filled rings or solid-filled circles that actually are closed polylines with width.

To create a donut, you specify its inside and outside diameters and its center. You can continue creating multiple copies with the same diameter by specifying different center points. To create solid-filled circles, specify an inside diameter of 0.



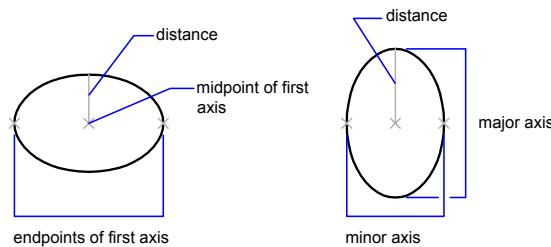
To create a donut

- 1 Click Draw menu ▶ Donut.
- 2 Specify the inside diameter (1).
- 3 Specify the outside diameter (2).
- 4 Specify the center of the donut (3).
- 5 Specify the center point for another donut, or press ENTER to complete the command.

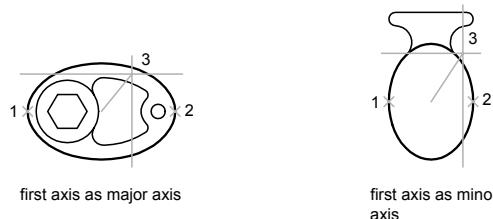


Draw Ellipses

The shape of an ellipse is determined by two axes that define its length and width. The longer axis is called the major axis, and the shorter one is the minor axis.



The illustrations below show two different ellipses created by specifying axis and distance. The third point specifies only a distance and does not necessarily designate the axis endpoint.



If you are drawing on isometric planes to simulate 3D, you can use ellipses to represent isometric circles viewed from an oblique angle. First you need to turn on Isometric Snap in the Drafting Settings dialog box.

See also:

"Draw Isometric Circles"
"Break and Join Objects" on page 652

To draw an isometric circle

- 1 Click Tools menu > Drafting Settings.
- 2 In the Drafting Settings dialog box, Snap and Grid tab, under Snap Type and Style, click Isometric Snap. Click OK.
- 3 Click Draw menu > Ellipse > Axis, End.
- 4 Enter **i** (Isocircle).
- 5 Specify the center of the circle.
- 6 Specify the radius or diameter of the circle.

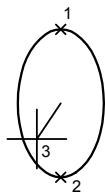
Draw toolbar



Command line: ELLIPSE

To draw a true ellipse using endpoints and distance

- 1 Click Draw menu > Ellipse > Axis, End.
- 2 Specify the first endpoint of the first axis (1).
- 3 Specify the second endpoint of the first axis (2).
- 4 Drag the pointing device away from the midpoint, and click to specify a distance (3) for half the length of the second axis.



Draw toolbar

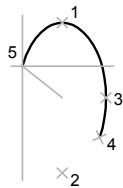


Command line: ELLIPSE

To draw an elliptical arc using start and end angles

- 1 Click Draw menu ► Ellipse ► Arc.
- 2 Specify endpoints for the first axis (1 and 2).
- 3 Specify a distance to define half the length of the second axis (3).
- 4 Specify the start angle (4).
- 5 Specify the end angle (5).

The elliptical arc is drawn counterclockwise between the start point and endpoint.



Draw toolbar

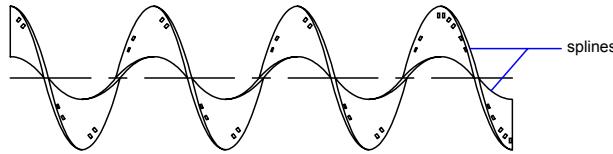


Command line: ELLIPSE

Draw Splines

A spline is a smooth curve that passes through or near a given set of points. You can control how closely the curve fits the points.

The SPLINE command creates a particular type of spline known as a nonuniform rational B-spline (NURBS) curve. A NURBS curve produces a smooth curve between control points.



You create splines by specifying points. You can close the spline so that the start and endpoints are coincident and tangent.

Tolerance describes how closely the spline fits the set of fit points you specify. The lower the tolerance, the more closely the spline fits the points. At zero tolerance, the spline passes through the points. You can change the spline-fitting tolerance while drawing the spline to see the effect.

You can use two methods for creating splines:

- Create spline curves with the Spline option of PEDIT to smooth existing polylines created with PLINE. Such spline-fit polylines are created with uniform knot vectors and are more likely to be included in drawings created with earlier versions of the product.
- Create splines, which are NURBS curves, with SPLINE. Drawings containing splines use less memory and disk space than those containing spline-fit polylines of similar shape.

You can easily convert spline-fit polylines into true splines with SPLINE.

See also:

- “Modify Splines” on page 669
- “Break and Join Objects” on page 652

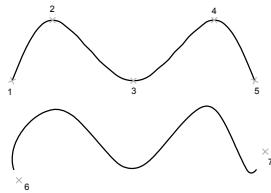
To convert a spline-fit polyline to a spline

- 1 Click Draw menu ► Spline.
- 2 Enter **o** (Object).
- 3 Select a spline-fit polyline and press ENTER.
The selected object changes from a polyline to a spline.

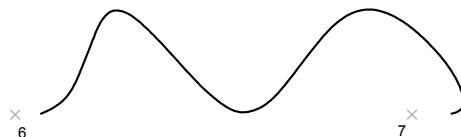
To convert a spline by specifying points

- 1 Click Draw menu ► Spline.
- 2 Specify the start point for the spline (1).

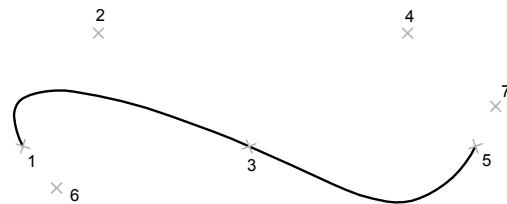
- Specify points (2 through 5) to create the spline, and press ENTER.
- Specify the start and end tangents (6, 7).



The spline below uses the same points but different start and end tangents.



The spline below uses the same points but a higher tolerance and different start and end tangents.



Draw toolbar



Command line: SPLINE

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 Helices” on page 672](#)

[Create a Solid or Surface by Sweeping](#)

To create a helix

- 1 Click Draw menu ► Helix.
- 2 Specify the center point for the base of the helix.
- 3 Specify the base radius.
- 4 Specify the top radius or press ENTER to specify the same value as the base radius.
- 5 Specify the height of the helix.

Modeling toolbar



Command line: HELIX

3D Make panel (click icon to expand), Helix

Draw Construction and Reference Geometry

Construction lines and reference points are temporary objects you create to help you draw accurately.

Draw Reference Points

Point objects are useful as nodes or reference geometry for object snaps and relative offsets.

You can set the style of the points and their size relative to the screen or in absolute units. Changing the style of points

- Makes them more visible and easier to differentiate from grid dots
- Affects the display of all point objects in the drawing
- Requires using REGEN to make the change visible

To set point style and size

- 1 Click Format menu ► Point Style.
- 2 In the Point Style dialog box, select a point style.
- 3 In the Point Size box, specify a size, either relative to the screen or in absolute units.
- 4 Click OK.

Command line: DDPTYPE

To create a point object

- 1 Click Draw menu ► Point ► Single Point.
- 2 Specify the point location.
You can snap to a point using the Node object snap.

Draw toolbar



Command line: POINT

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

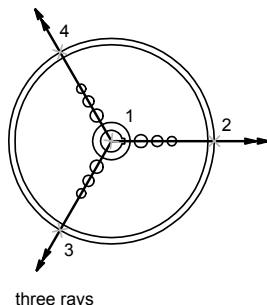
A construction line (xline) can be placed anywhere in three-dimensional space. You can specify its orientation in several ways. The default method for creating the line is the two-point method: you specify two points to define the orientation. The first point, the root, is the conceptual midpoint of the construction line, that is, the point snapped to by the Midpoint object snap.

You can also create construction lines in several other ways.

- **Horizontal and Vertical.** Create construction lines that pass through a point you specify and are parallel to the X or Y axis of the current UCS.
- **Angle.** Creates a construction line in one of two ways. Either you select a reference line and then specify the angle of the construction line from that line, or you create a construction line at a specific angle to the horizontal axis by specifying an angle and then a point through which the construction line should pass.
- **Bisector.** Creates a construction line that bisects an angle you specify. You specify the vertex and the lines that create the angle.
- **Offset.** Creates a construction line parallel to a baseline you specify. You specify the offset distance, select the baseline, and then indicate on which side of the baseline to locate the construction line.

Rays

A ray is a line in three-dimensional space that starts at a point you specify and extends to infinity. Unlike construction lines, which extend in two directions, rays extend in only one direction. Using rays instead of construction lines can help reduce visual clutter. Like construction lines, rays are ignored by commands that display the drawing extents.



To create a construction line by specifying two points

- 1 Click Draw menu ▶ Construction Line.
- 2 Specify a point to define the root of the construction line.
- 3 Specify a second point through which the construction line should pass.
- 4 Continue to specify construction lines as needed.
All subsequent xlines pass through the first point specified.
- 5 Press ENTER to end the command.

Draw toolbar



Command line: XLINE

To create a ray

- 1 Click Draw menu ▶ Ray.
- 2 Specify a starting point for the ray.
- 3 Specify a point through which the ray should pass.

- 4 Continue to specify points to create additional rays as needed.
All subsequent rays pass through the first point specified.
- 5 Press ENTER to end the command.

Command line: RAY

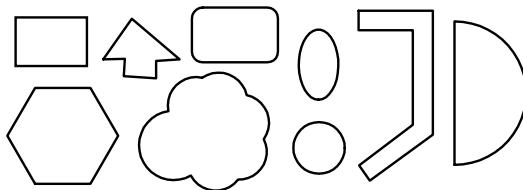
Create and Combine Areas (Regions)

Regions are two-dimensional enclosed areas that have physical properties such as centroids or centers of mass. You can combine existing regions into a single, complex region to calculate area.

Regions are two-dimensional enclosed areas you create from objects that form closed loops. Loops can be combinations of lines, polylines, circles, arcs, ellipses, elliptical arcs, and splines. The objects that make up the loops must either be closed or form closed areas by sharing endpoints with other objects.

Regions can be used for

- Applying hatching and shading
- Analyzing properties, such as area, using MASSPROP
- Extracting design information, such as the centroid



shapes that can form regions

You can create regions out of multiple loops and out of open curves whose endpoints are connected and form loops. You cannot form regions from open objects that intersect to form a closed area: for example, intersecting arcs or self-intersecting curves.

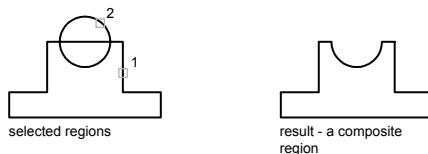
You can also create regions using BOUNDARY.

You create composite regions by combining, subtracting, or finding the intersection of regions. After forming these more complex regions, you can apply hatching or analyze their area.

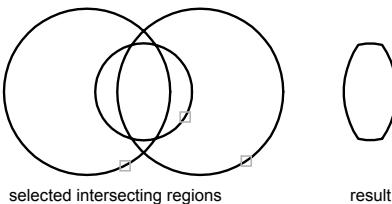
Objects combined using UNION:



Objects combined using SUBTRACT:



Objects combined using INTERSECT:



To define regions

- 1 Click Draw menu ► Region.
- 2 Select objects to create the region.
These objects must each form an enclosed area, such as a circle or a closed polyline.
- 3 Press ENTER.

A message on the command line indicates how many loops were detected and how many regions were created.

Command line: REGION

To define regions by using boundaries

- 1 Click Draw menu ► Boundary.
- 2 In the Boundary Creation dialog box, in the Object Type list, select Region.
- 3 Click Pick Points.
- 4 Specify a point in your drawing inside each closed area that you want to define as a region and press ENTER.

This point is known as the internal point.

NOTE You can make a new boundary set to limit the objects used to determine the boundary.

Command line: BOUNDARY

To combine regions by adding

- 1 Click Modify menu ► Solid Editing ► Union.
- 2 Select one region for the union.
- 3 Select another region.
You can select regions to unite in any order.
- 4 Continue selecting regions or press ENTER to end the command.
The command converts the selected regions to a new combined region.

Command line: UNION

To combine regions by subtracting

- 1 Click Modify menu ► Solid Editing ► Subtract.
- 2 Select one or more regions from which to subtract and press ENTER.
- 3 Select the region to subtract and press ENTER.
The areas of the second regions you selected are subtracted from the areas of the first regions.

Command line: SUBTRACT

To combine regions by finding intersections

- 1 Click Modify menu ► Solid Editing ► Intersect.

2 Select one region of the intersection.

3 Select another intersecting region.

You can select regions in any order to find their intersection.

4 Continue selecting regions or press ENTER to end the command.

The command converts the selected regions to a new region defined by the intersection of the selected regions.

Command line: INTERSECT

Create Revision Clouds

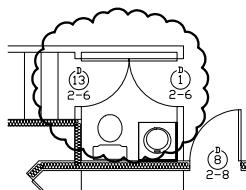
Revision clouds are polylines that consist of sequential arcs. They are used to call attention to parts of a drawing during the review stage.

If you review or redline drawings, you can increase your productivity by using the Revision Cloud feature to highlight your markups. REVCLLOUD 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.

REVCLLOUD stores the last used arc length as a multiple of the DIMSCALE system variable to provide consistency among drawings with different scale factors.



Make sure that you can see the entire area to be outlined with REVCLOUD before you begin the command. REVCLOUD is not designed to support transparent and real-time panning and zooming.

To create a revision cloud from scratch

- 1 Click Draw menu ► Revision Cloud.
- 2 At the Command prompt, specify a new minimum and maximum arc length or specify a revision cloud starting point.
The default minimum and maximum arc lengths are set to 0.5000 units. The maximum arc length can be no more than three times the minimum arc length.
- 3 Guide the crosshairs along the cloud path. You can click pick points along the path if you want to vary the size of the arcs.
- 4 Press ENTER at any time to stop drawing the revision cloud.
To close the revision cloud, return to its starting point.

Draw toolbar



Command line: REVCLOUD

To create revision clouds with a calligraphy pen style

- 1 Click Draw menu ► Revision Cloud.
- 2 At the Command prompt, enter **style**.
- 3 At the Command prompt, enter **calligraphy**.
- 4 Press ENTER to save the calligraphy setting and to continue with the command, or press ESC to end the command.

Draw toolbar



Command line: REVCLOUD

To convert an object to a revision cloud

- 1 Click Draw menu ► Revision Cloud.

- 2** At the Command prompt, specify a new minimum and maximum arc length or press ENTER.

The default minimum and maximum arc lengths are set to 0.5000 units. The maximum arc length can be no more than three times the minimum arc length.

- 3** Select the circle, ellipse, polyline, or spline that you want to convert to a revision cloud.

To reverse the direction of the arcs, enter **yes** on the command line and press ENTER.

- 4** Press ENTER to change the selected object to a revision cloud.

Draw toolbar



Command line: REVCLOUD

To change the default values for arc lengths in a revision cloud

- 1** Click Draw menu ► Revision Cloud.

- 2** At the Command prompt, specify a new minimum arc length and press ENTER.

- 3** At the Command prompt, specify a new maximum arc length and press ENTER.

The maximum arc length can be no more than three times the minimum arc length.

- 4** Press ENTER to continue with the command or ESC to end the command.

Draw toolbar



Command line: REVCLOUD

To edit the individual lengths of arcs or chords in a revision cloud

- 1** In your drawing, select the revision cloud you want to edit.

- 2** Move the pick points along the path of the revision cloud to change the arc lengths and chords.

20

Create and Use Blocks (Symbols)

A block is one or more objects combined to create a single object. Blocks help you reuse objects in the same drawing or in other drawings.

In this chapter

- Overview of Blocks
- Create and Store Blocks
- Add Dynamic Behavior to Blocks
- Control the Color and Linetype Properties in Blocks
- Nest Blocks
- Insert Blocks
- Work with Dynamic Blocks in Drawings
- Attach Data to Blocks (Block Attributes)
- Modify Blocks
- Change the Color and Linetype in a Block
- Disassemble a Block Reference (Explode)
- Remove Block Definitions

Overview of Blocks

You can use several methods to create blocks:

- Combine objects to create a block definition in your current drawing.
- Use the Block Editor to add dynamic behavior to a block definition in your current drawing.
- Create a drawing file and later insert it as a block in other drawings.
- Create a drawing file with several related block definitions to serve as a block library.

A block can be composed of objects drawn on several layers with various colors, linetypes, and linewidth 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.

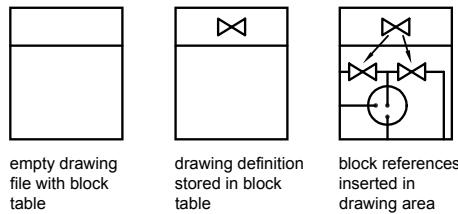
A block definition can also contain elements that add dynamic behavior to the block. You add these elements to the block in the Block Editor. When you add dynamic behavior to a block, you add flexibility and intelligence to the geometry. When you insert a block reference with dynamic behavior in a drawing, you can manipulate the geometry of the block reference through custom grips or custom properties, depending on how the block was defined.

You can use PURGE to remove unused block definitions from a drawing.

Create and Store Blocks

How Blocks Are Stored and Referenced

Every drawing file has an invisible data area called the block definition table. The block definition table stores all block definitions, which consist of all information associated with the block. It is these block definitions that are referenced when you insert blocks in your drawing. The following illustrations are conceptual representations of three drawing files. Each rectangle represents a separate drawing file and is divided into two parts: the smaller part represents the block definition table, and the larger part represents the objects in a drawing.



When you insert a block, you are inserting a block reference. The information is not simply copied from the block definition to the drawing area. Instead, a link is established between the block reference and the block definition. Therefore, if the block definition is changed, all references are updated automatically.

To reduce the size of a drawing, you can purge unused block definitions.

No procedures for this topic.

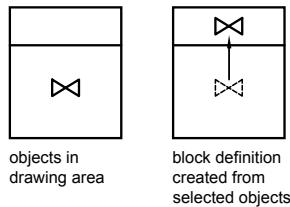
Create Blocks Within a Drawing

After you define a block in a drawing, you can insert a block reference in the drawing as many times as necessary. Use this method to create blocks quickly.

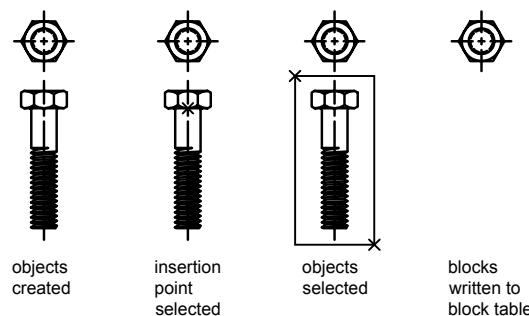
Each block definition includes a block name, one or more objects, the coordinate values of the base point to be used for inserting the block, and any associated attribute data.

The base point is used as a reference for positioning the block when you insert it. Suppose you specify that the base point is at the lower-left corner of an object in the block. Later, when you insert the block, you are prompted for an insertion point. The block base point is aligned at the insertion point you specified.

The block definition in the illustration comprises a name, PLUG_VALVE, four lines, and a base point at the intersection of the two diagonal lines. For an explanation of the schematic representation shown, see “Overview of Blocks” on page 460.



The illustration shows a typical sequence for creating a block definition within a drawing.



You can also use the Block Editor to create blocks that are saved within a drawing. For more information about using the Block Editor, see “Use the Block Editor” on page 471.

To define a block for the current drawing

- 1 Create the objects you want to use in the block definition.
- 2 On the Draw menu, click Block ► Make.
- 3 In the Block Definition dialog box, enter a block name in the Name box.
- 4 Under Objects, select Convert to Block.
If you want the original objects used to create the block definition to remain in your drawing, make sure the Delete option is not selected. If this option is selected, the original objects are erased from the drawing. If necessary, you can use OOPS to restore them.
- 5 Click Select Objects.

- 6 Use your pointing device to select the objects to be included in the block definition. Press ENTER to complete object selection.
- 7 In the Block Definition dialog box under Base Point, specify the block insertion point using one of these methods:
 - Click Pick Point to specify a point using the pointing device.
 - Enter the X,Y,Z coordinate values of the point.
- 8 In the Description box, enter a description for the block definition. This description is displayed in DesignCenter™ (ADCENTER).
- 9 Click OK.

The block is defined in the current drawing and can be inserted at any time.

Draw toolbar



Command line: BLOCK

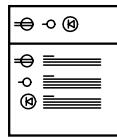
Create Block Libraries

A block library is a collection of block definitions stored in a single drawing file. You can use block libraries supplied by Autodesk or other vendors or create your own.

You can organize a set of related block definitions by creating the blocks in the same drawing file. Drawing files used this way are called block, or symbol, libraries. These block definitions can be inserted individually into any drawing that you are working on. Block library drawings are not different from other drawing files except in how they are used.

When you use BLOCK to define each block definition in the block library drawing, you can include a short description of the block that can be viewed in DesignCenter.

Optionally, you can also document each block definition by inserting it in the drawing area of the library drawing. In addition to the block geometry, you can include text that provides the block name, the date of creation, the date of the last modification, and any special instructions or conventions. This creates a visual index of the blocks in the block library drawing.



sample block
library drawing

Use DesignCenter to view and copy block definitions individually from block library drawings (or from any existing drawing) to your current drawing. DesignCenter does not overwrite an existing block definition in a drawing with one that comes from another drawing.

To create a block library drawing

- 1 Begin a new drawing.
- 2 Define a block.
- 3 Repeat step 2 for as many related block definitions as you want to make.
- 4 Save the drawing using a name appropriate for a library drawing.

These blocks can be inserted into any drawing using DesignCenter (ADCENTER).

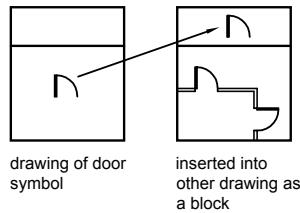
Draw toolbar



Command line: BLOCK

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 1077.

Use Paper Space Objects in Blocks

Objects in paper space are not included when you insert a drawing as a block. To transfer paper space objects to another drawing, make the objects into a block or save them in a separate drawing file, and then insert the block or drawing file into the other drawing.

To create a new drawing file from selected objects

- 1 Open an existing drawing or create a new drawing.
- 2 At the Command prompt, enter **wblock**.
- 3 In the Write Block dialog box, select Objects.
If you want the original objects used to create the new drawing to remain in your drawing, make sure the Delete From Drawing option is not selected. If this option is selected, the original objects are erased from the drawing. If necessary, you can use OOPS to restore them.
- 4 Click Select Objects.
- 5 Use your pointing device to select the objects to be included in the new drawing. Press ENTER to complete object selection.
- 6 In the Write Block dialog box under Base Point, specify the point to be the origin point (0,0,0) for the new drawing using one of these methods:
 - Click Specify Point to specify a point using the pointing device.
 - Enter the X,Y,Z coordinate values of the point.
- 7 Under Destination, enter a file name and path for the new drawing, or click the [...] button to display a standard file selection dialog box.
- 8 Click OK.

A new drawing is created with the selected objects.

Command line: WBLOCK

To create a new drawing file from an existing block definition

- 1 On the Modify menu, click Object ► Block Description.
- 2 In the Block Definition dialog box, in the Name box, select the block to modify.
- 3 In the Name box, enter a new name.
- 4 In the Description box, enter or modify the description for the new drawing file.
- 5 Click OK.

Command line: BLOCK

Use Tool Palettes to Organize Blocks

You can use tool palettes to organize blocks that are stored in one drawing file or separate drawing files.

Once you've added a block tool to a tool palette, you can easily insert the block reference in your drawing by dragging it from the tool palette to the drawing or by clicking and placing it in the drawing. For information about using tool palettes to organize and insert blocks, see Create and Use Tools from Objects and Images.

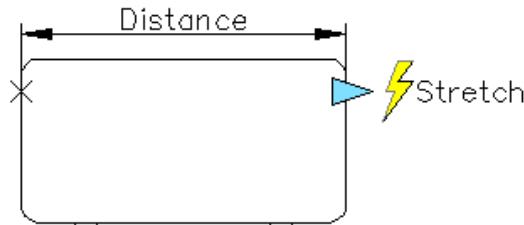
No procedures for this topic.

Add Dynamic Behavior to Blocks

When you add dynamic behavior to a block definition, you add flexibility and intelligence to the block geometry. Instead of being a fixed part of a drawing, a dynamic block reference can be changed or manipulated as you work in a drawing.

Quick Start to Creating Dynamic Blocks

You add dynamic behavior to new or existing block definitions by adding parameters and actions to the block in the Block Editor. In the following example, a desk block is shown in the Block Editor. The block contains a linear parameter, which displays similar to a dimension and is labeled "Distance," and a stretch action, which displays a lightning bolt and a "Stretch" label.



For a block to be dynamic, you must add at least one parameter. You then add an action and associate the action with the parameter. The types of parameters and actions you add to the block definition define how the block reference will work in a drawing. To see demonstrations of how parameters and actions are added to a block, see the New Features Workshop. Click Help ► New Features Workshop ► Produce ► Dynamic Blocks.

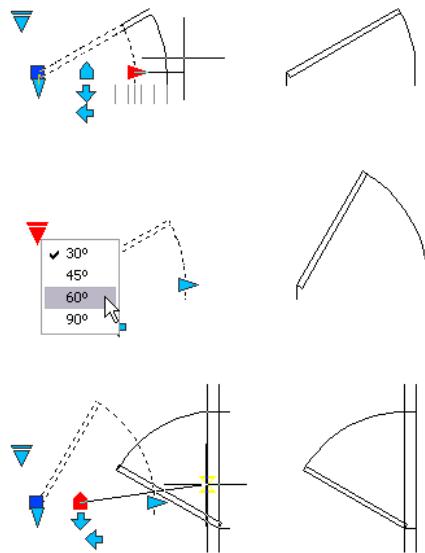
For more information about creating dynamic blocks, see:

- “Overview of Dynamic Blocks” on page 468
- “Overview of Dynamic Block Elements” on page 482

Overview of Dynamic Blocks

A *dynamic* block has flexibility and intelligence. A dynamic block reference can easily be changed in a drawing while you work. You can manipulate the geometry in a dynamic block reference through custom grips or custom properties. This allows you to adjust the block in-place as necessary rather than searching for another block to insert or redefining the existing one.

For example, if you insert a door block reference in a drawing, you might need to change the size of the door while you're editing the drawing. If the block is dynamic and defined to have an adjustable size, you can change the size of the door simply by dragging the custom grip or by specifying a different size in the Properties palette. You might also need to change the open angle of the door. The door block might also contain an alignment grip, which allows you to align the door block reference easily to other geometry in the drawing.



You use the Block Editor to create dynamic blocks. The Block Editor is a special authoring area where you add the elements that make a block dynamic. You can create a block from scratch, or you can add dynamic behavior to an existing

block definition. You can also create geometry, just as you would in the drawing area.

You add *parameters* and *actions* to a block to make it dynamic. When you add these elements to the block, you add flexibility and intelligence to the block geometry.

- Parameters define custom properties for the dynamic block by specifying positions, distances, and angles for geometry in the block.
- Actions define how the geometry of a dynamic block reference will move or change when the block reference is manipulated in a drawing. When you add actions to the block, you must associate them with parameters and usually geometry.

When you add a parameter to a block definition, custom grips and properties are automatically added to the block. You use these custom grips and properties to manipulate the block reference in the drawing.

Process for Creating Dynamic Blocks

To create quality dynamic blocks, so you get the results you expect, it is recommended that you follow the steps in the following process. This process will help you author dynamic blocks effectively.

Step 1. Plan the contents of the dynamic block before you create it

Before you create a dynamic block, you should know what it will look like and how it will be used in a drawing. Decide which objects within the block will change or move when the dynamic block reference is manipulated. Furthermore, decide *how* these objects will change. For example, you might create a dynamic block that can be resized. In addition, when the block reference is resized, additional geometry may be displayed. These factors determine the types of parameters and actions you add to the block definition, and how you make the parameters, actions, and geometry work together.

Step 2. Draw the geometry

You can draw the geometry for your dynamic block in the drawing area or the Block Editor. You can also use existing geometry in a drawing or an existing block definition.

NOTE If you will use visibility states to change how geometry is displayed in the dynamic block references, you may not want to include all the geometry at this point. For more information about working with visibility states, see “Create Visibility States” on page 523.

Step 3. Understand how the block elements will work together

Before you add parameters and actions to your block definition, understand their *dependencies* on each other and on the geometry within the block. When you add an action to the block definition, you will need to associate the action with a parameter and a selection set of geometry. This creates a dependency. When you add multiple parameters and actions to a dynamic block reference, you will need to set up the correct dependencies in order for the block reference to function properly in a drawing.

For example, you create a dynamic block that contains several objects. Some of the objects have a stretch action associated with them. You also want all the objects to rotate around the same base point. In this case, you should add the rotate action after adding all the other parameters and actions. If the rotate action isn't associated with all of the other objects (geometry, parameters, and actions) in the block definition, parts of the block reference may not rotate or manipulating the block reference may cause unexpected results.

Step 4. Add parameters

Add the appropriate parameters to the dynamic block definition, following the prompts on the command line. For more information about using parameters, see “Use Parameters in Dynamic Blocks” on page 484.

NOTE You can use the Parameter Sets tab of the Block Authoring Palettes to add a parameter and an associated action at the same time. For more information about using parameter sets, see “Use Parameter Sets” on page 518.

Step 5. Add actions

Add the appropriate actions to the dynamic block definition. Follow the prompts on the command line, making sure to associate the actions with the correct parameters and geometry. For more information about using actions, see “Overview of Using Actions in Dynamic Blocks” on page 496.

Step 6. Define how the dynamic block reference will be manipulated

You can specify how the dynamic block reference will be manipulated in the drawing. You can manipulate a dynamic block reference through custom grips and custom properties. When you create a dynamic block definition, you define what grips are displayed and how they edit the dynamic block reference. You also specify whether or not custom properties for the block will be shown in the Properties palette and whether or not these properties can be changed through the palette or through custom grips.

Step 7. Save the block and then try it in a drawing

Save the dynamic block definition and exit the Block Editor. Insert the dynamic block reference in a drawing and test the block's functionality.

See also:

["Save a Block in the Block Editor" on page 551](#)

No procedures for this topic.

Use the Block Editor

You use the Block Editor to add dynamic behavior to blocks. The Block Editor provides all the tools you need to add intelligence and flexibility to your blocks.

Overview of the Block Editor

The Block Editor is a special authoring area for creating block definitions and adding dynamic behavior.

The Block Editor provides special Authoring palettes. These palettes provide quick access to block authoring tools.

In addition to the Block Authoring palettes, the Block Editor provides a drawing area in which you can draw and edit geometry as you would in the program's main drawing area. You can specify the background color for the Block Editor drawing area.

NOTE You can use most commands in the Block Editor. When you enter a command that is not allowed in the Block Editor, a message is displayed on the command line.

You can use the Block Editor to edit or add dynamic behavior to block definitions that exist in the current drawing. You can also use it to create new block definitions.

In the Block Editor, a special toolbar is displayed above the drawing area. The toolbar shows the name of the block definition currently being edited and provides tools to do the following:

- Save the block definition
- Add a parameter
- Add an action
- Define attributes

- Close the Block Editor
- Manage visibility states

You can select any parameter, grip, action, or geometric object in the Block Editor to view its properties in the Properties palette. When you select an object in the Block Editor, the coordinate values shown in the Properties palette reflect the block definition space.

When you work in the Block Editor, the command line should be displayed. The command line displays prompts for nearly all aspects of creating dynamic blocks.

UCS in the Block Editor

A UCS icon is displayed in the drawing area of the Block Editor. The origin of the UCS icon defines the base point for the block. You can change the base point for the block by moving the geometry relative to the origin of the UCS icon, or by adding a base point parameter.

The UCS command is disabled in the Block Editor. You can open an existing 3D block definition in the Block Editor and assign parameters to the block. However, the parameters will ignore any Z coordinate values in the block space. Consequently, the block reference cannot be edited along the Z axis. Furthermore, while you can create a dynamic block that contains solid objects and add actions to it, such as move, rotate, and scale, you can't perform solid editing features within a dynamic block reference (for example, stretch a solid, move a hole within a solid, and so on).

See also:

Define Block Attributes

To open an existing block definition in the Block Editor

- 1 Click Tools menu ► Block Editor.
- 2 In the Edit Block Definition dialog box, do one of the following:
 - Select a block definition from the list.
 - Select <Current Drawing> if the drawing is the block definition you want to open.
- 3 Click OK.

Standard toolbar



Command line: BEDIT

Shortcut menu: Right-click a selected block. Click Block Editor.

To create a new block definition in the Block Editor

- 1 Click Tools menu ▶ Block Editor.
- 2 In the Edit Block Definition dialog box, enter a name for the new block definition.
- 3 Click OK.
- 4 In the Block Editor, click Save Block Definition.

NOTE This saves the block definition even if you have not added any objects in the Block Editor drawing area.

- 5 Click Close Block Editor.

Standard toolbar



Command line: BEDIT

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

- 1 If the Tool Palettes window is not already open, click Tools menu ▶ Palettes ▶ Tool Palettes.
- 2 Right-click a block icon.
- 3 Click Block Editor.

NOTE A block on a tool palette may reside in another drawing. The drawing that contains the block definition is opened in the Block Editor.

Standard toolbar



Command line: TOOLPALETTES

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

- 1 If DesignCenter is not already open, click Tools menu ► Palettes ► DesignCenter.
- 2 Right-click a block icon.
- 3 Click Block Editor.

Standard toolbar



Command line: ADCENTER

To open a drawing file saved as a block (not dynamic) in the Block Editor

- 1 Open the drawing file that is saved as a block.
- 2 Click Tools menu ► Block Editor.
- 3 In the Edit Block Definition dialog box, select <Current Drawing>.
- 4 Click OK.

Standard toolbar



Command line: OPEN

To open a drawing file saved as a dynamic block in the Block Editor

- 1 Open the drawing file that is saved as a block.
An alert is displayed that states that the drawing file contains authoring elements.
- 2 In the alert dialog box, click Yes to open the drawing in the Block Editor.

Standard toolbar



Command line: OPEN

To view properties of a block definition in the Block Editor

- 1 Click Tools menu ► Block Editor.
- 2 In the Edit Block Definition dialog box, do one of the following:
 - Select a block definition from the list.
 - Select <Current Drawing> if the drawing is the block definition you want to open.
- 3 Click OK.
- 4 Make sure the Properties Palette window is displayed. If it's not displayed, click Tools menu ► Palettes ► Properties.
- 5 In the Properties Palette window, under Block, view the properties of the block definition.

Standard toolbar



Command line: BEDIT

To view properties of objects in the Block Editor

- 1 In the Block Editor, select an object.
- 2 Make sure the Properties Palette window is displayed. If it's not displayed, click Tools menu ► Palettes ► Properties.
- 3 In the Properties Palette window, view the properties of the selected object.

Standard toolbar



Command line: BEDIT

Shortcut menu: Right-click the selected object. Click Properties.

To close the Block Editor

- In the Block Editor, click Close Block Editor.

Block Editor toolbar

Close Block Editor

Command line: BCLOSE

To prevent access to the Block Editor

- On the command line, enter **blockedlock**.
- Enter **1**, and then press ENTER.

Use Block Authoring Palettes

The Block Editor has three block authoring palettes: Parameters, Actions, and Parameter Sets.

The Block Authoring Palettes window is displayed only in the Block Editor. Use these palettes to add parameters and actions to your dynamic block definition.

Create Custom Block Authoring Tools

You can create custom block authoring tools. In order to preserve the default tools on the block authoring palettes, you should create a new palette for custom block authoring tools. You can then copy a parameter set tool from one of the existing palettes and paste the copy onto the new palette. In the Tool Properties Dialog Box, you can then change the properties of the new tool, including the following:

- Tool description
- Parameter type
- Associated action(s)
- Key point on the parameter to which the action is tied (if applicable)
- Tool palette image

You cannot drag parameters and actions from the Block Editor onto any tool palette.

See also:

- “Use Parameters in Dynamic Blocks” on page 484
- “Use Actions with Dynamic Blocks” on page 496
- “Use Parameter Sets” on page 518
- “Control Tool Properties” on page 90
- “Organize Tool Palettes” on page 98

To show or hide the Block Authoring palettes in the Block Editor

- In the Block Editor, on the Block Editor toolbar, click the Block Authoring Palettes button.

Block Editor toolbar



Command line: BAUTHORPALETTE, BAUTHORPALETTECLOSE

To make a copy of a parameter set

- 1 In the Block Editor, in the Block Authoring Palettes window, Parameters Sets tab, right-click a parameter set. Click Copy.
- 2 Right-click anywhere on the palette to which you want to add the parameter set (except on a parameter set). Click Paste.

To add an action to a parameter set

- 1 In the Block Editor, in the Block Authoring Palettes window, Parameters Sets tab, right-click a parameter set. Click Properties.
- 2 In the Tool Properties dialog box, under Parameter, click Actions, and then click the [...] button.
- 3 In the Add Actions dialog box, under Action Object to Add, select an action from the list.
- 4 Click Add.
- 5 (Optional) Repeat steps 3 and 4 to add additional actions.
- 6 Click OK.
- 7 In the Tool Properties dialog box, click OK.

To delete an action from a parameter or parameter set

- 1 In the Block Editor, in the Block Authoring Palettes window, right-click a parameter or parameter set. Click Properties.
- 2 In the Tool Properties dialog box, under Parameter, click Actions, and then click the [...] button.
- 3 In the Add Actions dialog box, under Action Object List, select an action from the list.

- 4** Click Delete.
- 5** (Optional) Repeat steps 3 and 4 to delete additional actions.
- 6** Click OK.
- 7** In the Tool Properties dialog box, click OK.

How Objects Are Displayed in the Block Editor

Parameters, actions, and their relationships (dependencies) are displayed in different ways in the Block Editor. You can specify settings for some of these elements.

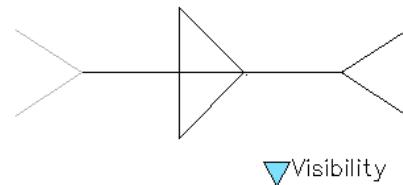
Parameters

In the Block Editor, most parameters look like dimensions. If you create a value set (a range or list of values) for a parameter, tick marks are shown at the locations of those values.

You can specify the following settings for parameters in the Block Editor:

- Parameter color
- Parameter text and arrow size
- Parameter font
- Grip color
- Display of value set markers (tick marks) for parameters

When you use a visibility parameter in your dynamic block definition, you can specify which geometric objects are invisible for a given visibility state. You can specify whether or not geometry that is made invisible for visibility states displays in the Block Editor. In the following example, a visibility state is displayed in the Block Editor. The geometry that displays in a dimmed state is invisible for that visibility state.



Actions

An action displays its name and icon (a lightning bolt) in the Block Editor. You can specify the text size and color for actions in the Block Editor.

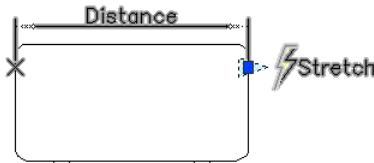
Grips

You can specify grip size and color for display in the Block Editor. This setting does not affect the size and color of the grips in a block reference in a drawing.

Dependencies

When you select a parameter, grip, or action in the Block Editor, its associated objects, or dependencies, are highlighted. This is called *dependency highlighting*. You can turn dependency highlighting on or off.

The following example shows how dependency highlighting creates a halo effect for the associated parameter (labeled Distance) and action (labeled Stretch) when you select the custom grip in the Block Editor.



The following table details what is dependency highlighted when you select an element in the Block Editor.

Selected object in the Block Editor	Objects that are dependency highlighted
Parameter	Associated grips and actions
Grip	Associated parameter and actions
Action	Associated parameters, grips, and the selection set (geometry)

To specify the display color for parameters in the Block Editor

- 1 On the command line, enter **bparametercolor**.
- 2 Enter one of the following values:
 - **BYLAYER**
 - **BYBLOCK**
 - An integer from 1 to 255

- A true color specified by three integers each ranging from 1 to 255 in the following format: **RGB:000,000,000**.

3 Press ENTER.

To specify the text size for parameters and actions in the Block Editor

- 1 On the command line, enter **bparametersize**.
- 2 Enter an integer from 1 to 255 (pixels).
- 3 Press ENTER.

To specify the font for parameters in the Block Editor

- 1 On the command line, enter **bparameterfont**.
- 2 Enter any TrueType or SHX font on your system.
- 3 Press ENTER.

To specify the text color for actions in the Block Editor

- 1 On the command line, enter **bactioncolor**.
- 2 Enter one of the following values:
 - **BYLAYER**
 - **BYBLOCK**
 - An integer from 1 to 255.
 - A true color specified by three integers each ranging from 1 to 255 in the following format: **RGB:000,000,000**.
- 3 Press ENTER.

To specify the display size of grips in the Block Editor

- 1 On the command line, enter **bgripobjsize**.
- 2 Enter an integer from 1 to 255 (pixels).
- 3 Press ENTER.

To specify the display color of grips in the Block Editor

- 1 On the command line, enter **bgripobjcolor**.

2 Enter one of the following values:

■ **BYLAYER**

■ **BYBLOCK**

■ An integer from 1 to 255.

■ A true color specified by three integers each ranging from 1 to 255 in the following format: **RGB:000,000,000**.

3 Press ENTER.

To update the text and grip display size at their specified values in the Block Editor

1 In the Block Editor, on the Block Editor toolbar, click Update Parameter and Action Text Size.

NOTE When you zoom in and out in the Block Editor, text and grip size will change relative to the zoom factor. However, you can update the Block Editor to display the text and grip size at their specified values.

Command line: REGEN

To specify whether or not objects that are invisible for a visibility state can be seen in the Block Editor

1 On the command line, enter **bvmode**.

2 Enter one of the following values:

■ **0** Specifies that objects that are invisible are not visible in the Block Editor.

■ **1** Specifies that objects that are invisible are dimmed in the Block Editor.

3 Press ENTER.

Block Editor toolbar



To turn dependency highlighting on or off in the Block Editor

1 On the command line, enter **bdependencyhighlight**.

- 2** Enter one of the following values:
 - **0** Turns off dependency highlighting.
 - **1** Turns on dependency highlighting.
- 3** Press ENTER.

To control whether value set markers (tick marks) are displayed

- 1** On the command line, enter **btmarkdisplay**.
- 2** Enter one of the following values:
 - **0** Specifies that value set markers are not displayed.
 - **1** Specifies that value set markers are displayed.
- 3** Press ENTER.

Add Dynamic Elements to Blocks

To make a block dynamic, you need to add specific elements to the block definition.

Overview of Dynamic Block Elements

You add dynamic elements to a block definition in the Block Editor. In addition to geometry, a dynamic block generally includes one or more parameters and one or more actions.

- **Parameters.** Define custom properties for the dynamic block by specifying positions, distances, and angles for geometry in the block.
- **Actions.** Define how the geometry of a dynamic block reference will move or change when you manipulate the block reference in a drawing. When you add actions to a dynamic block definition, you must associate the actions with parameters. You also specify a selection set of geometry that the action will affect.

NOTE Parameters and actions are displayed only in the Block Editor. When you insert a dynamic block reference in a drawing, the parameters and actions contained in the dynamic block definition are not displayed.

When you add a parameter to a dynamic block definition, grips are added to key points of the parameter. *Key points* are the parts of a parameter that you

use to manipulate the block reference. For example, a linear parameter has key points at its base point and end point. You can manipulate the parameter distance from either key point.

The type of parameter that you add to a dynamic block determines the type of grips that are added. Each type of parameter supports only certain types of actions. The following table shows the relationship among parameters, grips, and actions.

Parameter Type	Grip Type	Actions You Can Associate with a Parameter
Point		Move, Stretch
Linear		Move, Scale, Stretch, Array
Polar		Move, Scale, Stretch, Polar Stretch, Array,
XY		Move, Scale, Stretch, Array
Rotation		Rotate
Flip		Flip
Alignment		None (The action is implied and contained within the parameter.)
Visibility		None (The action is implied and controlled by visibility states.)
Lookup		Lookup
Base		None

To create a dynamic block

- 1 Click Tools menu ► Block Editor.
- 2 In the Edit Block Definition dialog box, do one of the following:
 - Select a block definition from the list.
 - Select <Current Drawing> if you want to save the drawing as a dynamic block.
 - Under Block to Create or Edit, enter a name for a new block definition.
- 3 Click OK.
- 4 In the Block Editor, add or edit geometry as necessary.
- 5 Do one of the following:
 - Add one or more parameter sets from the Parameter Sets tab of the Block Authoring Palettes, following the prompts on the command line. Double-click the yellow alert icon (or use the BACTIONSET command) and follow the prompts on the command line to associate the action with a selection set of geometry.
 - Add one or more parameters from the Parameters tab of the Block Authoring Palettes, following the prompts on the command line. Add one or more actions from the Actions tab, following the prompts on the command line.
- 6 On the Block Editor toolbar, click Save Block Definition.
- 7 Click Close Block Editor.

Standard toolbar



Command line: BEDIT

Shortcut menu: Right-click a block. Click Block Editor.

Use Parameters in Dynamic Blocks

You add parameters to a dynamic block definition in the Block Editor. In the Block Editor, parameters have an appearance similar to dimensions. Parameters define custom properties for the block. Parameters also specify positions, distances, and angles for geometry in the block reference. When you add a

parameter to a dynamic block definition, the parameter defines one or more custom properties for the block.

For example, when you add a rotation parameter to a dynamic block definition, the rotation parameter defines an *Angle* property for the block reference. So, if you have a block of a chair in your drawing, and you want to be able to rotate the block's position as you edit, the parameter defines the axis whereby the block can be rotated.

If you add a point parameter to a dynamic block definition, the point parameter defines two custom properties for the block reference: *Position X* and *Position Y* (relative to the base point of the block reference.)

A dynamic block definition must contain at least one parameter. When a parameter is added to a dynamic block definition, grips associated with key points of the parameter are automatically added. You must then add an action to the block definition and associate the action with a parameter.

In a drawing, you use a grip or custom property in the Properties palette to manipulate the block reference. When you manipulate the block reference in a drawing, by moving a grip or changing the value of a custom property in the Properties palette, you change the value of the parameter that defines that custom property in the block. When you change the value of the parameter, it drives the action that is associated with that parameter, which changes the geometry (or a property) of the dynamic block reference.

Parameters also define and constrain values that affect the dynamic block reference's behavior in a drawing. Some parameters can have a fixed set of values, minimum and maximum values, or increment values. For example, a linear parameter used in a window block may have the following fixed set of values: 10, 20, 30, and 40. When the block reference is inserted in a drawing, you can only change the window to one of these values. Adding a value set to a parameter allows you to limit how the block reference is manipulated in a drawing. For more information about parameter value sets, see "Specify Value Sets for Dynamic Blocks" on page 542.

The point, linear, polar, XY, and rotation parameters have a property called Chain Actions. For more information about allowing chained actions for a parameter, see Allow Chained Actions for Dynamic Blocks.

You can also extract parameter values through the Attribute Extraction wizard or through an attribute extraction template file.

The following table lists and describes the types of parameters you can add to a dynamic block definition and the types of actions you can associate with each parameter.

Parameter Type	Description	Supported Actions
Point	Defines an X and Y location in the drawing. In the Block Editor, looks similar to an ordinate dimension.	Move, Stretch
Linear	Shows the distance between two anchor points. Constrains grip movement along a preset angle. In the Block Editor, looks similar to an aligned dimension.	Move, Scale, Stretch, Array
Polar	Shows the distance between two anchor points and displays an angle value. You can use both grips and the Properties palette to change both the distance value and the angle. In the Block Editor, looks similar to an aligned dimension	Move, Scale, Stretch, Polar
XY	Shows the X and Y distances from the base point of the parameter. In the Block Editor, displays as a pair of dimensions (horizontal and vertical).	Move, Scale, Stretch, Array
Rotation	Defines an angle. In the Block Editor, displays as a circle.	Rotate
Flip	Flips objects. In the Block Editor, displays as a reflection line. Objects can be flipped about this reflection line. Displays a value that shows if the block reference has been flipped or not.	Flip
Alignment	Defines an X and Y location and an angle. An alignment parameter always applies to the entire block and needs no action associated with it. An alignment parameter allows the block reference to automatically rotate around a point to align with another object in the drawing. An alignment parameter affects the rotation property of the block reference. In the Block Editor, looks like an alignment line.	None (The action is implied and contained within the parameter.)
Visibility	Controls the visibility of objects in the block. A visibility parameter always applies to the entire block and needs no action associated with it. In a drawing, you click the grip to display a list of visibility states available for the block reference. In the Block Editor, displays as text with an associated grip.	None (The action is implied and controlled by visibility states.)
Lookup	Defines a custom property that you can specify or set to evaluate a value from a list or table you define. It can be associated with a single lookup grip. In the block reference, you click the grip to display a list of available values. In the Block Editor, displays as text with an associated grip.	Lookup
Base	Defines a base point for the dynamic block reference relative to the geometry in the block. Cannot be associated with any actions, but can belong to an action's	None

Parameter Type	Description	Supported Actions
	selection set. In the Block Editor, displays as a circle with crosshairs.	

See also:

- “Specify Value Sets for Dynamic Blocks” on page 542
- “Overview of Specifying Custom Properties for Dynamic Blocks” on page 539
- “Extract Data from Block Attributes” on page 566
- “Extract Block Attribute Data (Advanced)” on page 568

To add a point parameter to a dynamic block definition

- 1 In the Block Editor, in the Block Authoring Palettes window, Parameters tab, click the Point Parameter tool.
- 2 (Optional) Follow the prompts on the command line to specify the following for the parameter:
 - Name
 - Label
 - Description
 - Chained Actions
 - Properties displayed for block reference

NOTE You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.

- 3 Do one of the following to specify a location for the parameter:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.
- 4 Do one of the following to specify a location for the parameter label:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.

The yellow alert icon indicates that you should associate an action with the parameter you just added.
- 5 To add an action now, double-click the alert icon. Follow the prompts to associate an action with the parameter and a selection set of geometry.

- 6 On the Block Editor toolbar, click Save Block Definition.
- 7 If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar



Command line: BPARAMETER

To add a linear parameter to a dynamic block definition

- 1 In the Block Editor, in the Block Authoring Palettes window, Parameters tab, click the Linear Parameter tool.
- 2 (Optional) Follow the prompts on the command line to specify the following for the parameter:
 - Name
 - Label
 - Description
 - Value set
 - Chained Actions
 - Properties displayed for block reference

NOTE You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.

- 3 Do one of the following to specify a base point for the parameter:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.
- 4 Do one of the following to specify an endpoint for the parameter:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.
- 5 Do one of the following to specify a location for the parameter label:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.

The yellow alert icon indicates that you should associate an action with the parameter you just added.

- 6 To add an action now, double-click the alert icon. Follow the prompts to associate an action with the parameter and a selection set of geometry.
- 7 On the Block Editor toolbar, click Save Block Definition.
- 8 If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar



Command line: BPARAMETER

To add a polar parameter to a dynamic block definition

- 1 In the Block Editor, in the Block Authoring Palettes window, Parameters tab, click the Polar Parameter tool.
- 2 (Optional) Follow the prompts on the command line to specify the following for the parameter:
 - Name
 - Label
 - Description
 - Value set
 - Chained Actions
 - Properties displayed for block reference

NOTE You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.

- 3 Do one of the following to specify a base point for the parameter:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.
- 4 Do one of the following to specify an endpoint for the parameter:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.

- 5 Do one of the following to specify a location for the parameter label:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.The yellow alert icon indicates that you should associate an action with the parameter you just added.
- 6 To add an action now, double-click the alert icon. Follow the prompts to associate an action with the parameter and a selection set of geometry.
- 7 On the Block Editor toolbar, click Save Block Definition.
- 8 If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar



Command line: BPARAMETER

To add an XY parameter to a dynamic block definition

- 1 In the Block Editor, in the Block Authoring Palettes window, Parameters tab, click the XY Parameter tool.
- 2 (Optional) Follow the prompts on the command line to specify the following for the parameter:
 - Name
 - Label
 - Description
 - Value set
 - Chained Actions
 - Properties displayed for block reference

NOTE You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.

- 3 Do one of the following to specify a base point for the parameter:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.

- 4 Do one of the following to specify an endpoint for the parameter:
 - Use the pointing device.
 - Enter the *X* and *Y* coordinate values, separated by a comma.
- The yellow alert icon indicates that you should associate an action with the parameter you just added.
- 5 To add an action now, double-click the alert icon. Follow the prompts to associate an action with the parameter and a selection set of geometry.
- 6 On the Block Editor toolbar, click Save Block Definition.
- 7 If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar



Command line: BPARAMETER

To add a rotation parameter to a dynamic block definition

- 1 In the Block Editor, in the Block Authoring Palettes window, Parameters tab, click the Rotation Parameter tool.
- 2 (Optional) Follow the prompts on the command line to specify the following for the parameter:
 - Name
 - Label
 - Description
 - Value set
 - Chained Actions
 - Properties displayed for block reference

NOTE You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.

- 3 Do one of the following to specify a base point for the parameter:
 - Use the pointing device.
 - Enter the *X* and *Y* coordinate values, separated by a comma.

- 4** Do one of the following to specify the radius for the parameter:
 - Use the pointing device.
 - Enter a value.
- 5** Specify a base angle for the parameter.

The angle shown in the Properties palette when you select the block reference in a drawing is measured relative to the base angle specified in the block definition.
- 6** Do one of the following to specify a default rotation angle for the parameter:
 - Use the pointing device.
 - Enter an angle value.
- 7** Do one of the following to specify a location for the parameter label:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.

The yellow alert icon indicates that you should associate an action with the parameter you just added.
- 8** To add an action now, double-click the alert icon. Follow the prompts to associate an action with the parameter and a selection set of geometry.
- 9** On the Block Editor toolbar, click Save Block Definition.
- 10** If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar



Command line: BPARAMETER

To add an alignment parameter to a dynamic block definition

- 1** In the Block Editor, in the Block Authoring Palettes window, Parameters tab, click the Alignment Parameter tool.
- 2** Follow the prompts on the command line to specify the parameter name.
- 3** Do one of the following to specify the base point of the alignment parameter:
 - Use the pointing device.

- Enter the X and Y coordinate values, separated by a comma.

This base point is the default location for the alignment parameter grip.

- 4 (Optional) Enter **type** on the command line and follow the prompts to specify whether the parameter type is perpendicular or tangent.
- 5 Do one of the following to specify the alignment direction:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.
- 6 On the Block Editor toolbar, click Save Block Definition.
- 7 If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar



Command line: BPARAMETER

To add a flip parameter to a dynamic block definition

- 1 In the Block Editor, in the Block Authoring Palettes window, Parameters tab, click the Flip Parameter tool.
- 2 (Optional) Follow the prompts on the command line to specify the following for the parameter:
 - Name
 - Label
 - Description
 - Properties displayed for block reference

NOTE You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.

- 3 Do one of the following to specify the first point of the reflection line for the flip parameter:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.

This first point of the reflection line is the default location for the flip parameter grip.

- 4** Do one of the following to specify the second point of the reflection line for the flip parameter:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.
 - 5** Do one of the following to specify a location for the parameter label:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.
- The yellow alert icon indicates that you should associate an action with the parameter you just added.
- 6** To add an action now, double-click the alert icon. Follow the prompts to associate an action with the parameter and a selection set of geometry.
 - 7** On the Block Editor toolbar, click Save Block Definition.
 - 8** If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar



Command line: BPARAMETER

To add a visibility parameter to a dynamic block definition

- 1** In the Block Editor, in the Block Authoring Palettes window, Parameters tab, click the Visibility Parameter tool.
- 2** (Optional) Follow the prompts on the command line to specify the following for the parameter:
 - Name
 - Label
 - Description
 - Properties displayed for block reference

NOTE You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.

- 3** Do one of the following to specify a location for the parameter:
 - Use the pointing device.

- Enter the X and Y coordinate values, separated by a comma.

This is the location for the visibility parameter grip in the block reference.

- 4 On the Block Editor toolbar, click Save Block Definition.
- 5 If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar



Command line: BPARAMETER

To add a lookup parameter to a dynamic block definition

- 1 In the Block Editor, in the Block Authoring Palettes window, Parameters tab, click the Lookup Parameter tool.
- 2 (Optional) Follow the prompts on the command line to specify the following for the parameter:
 - Name
 - Label
 - Description
 - Properties displayed for block reference

NOTE You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.

- 3 Do one of the following to specify a location for the parameter:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.
The yellow alert icon indicates that you should associate an action with the parameter you just added.
- 4 To add an action now, double-click the alert icon. Follow the prompts to associate a lookup action with the parameter. This will display the Property Lookup Table dialog box, which you can complete now or later.
- 5 Click OK.
- 6 On the Block Editor toolbar, click Save Block Definition.
- 7 If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar



Command line: BPARAMETER

To add a base point parameter to a dynamic block definition

- 1 In the Block Editor, in the Block Authoring Palettes window, Parameters tab, click the Base Point Parameter tool.
- 2 Do one of the following to specify a location for the parameter:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.
- 3 On the Block Editor toolbar, click Save Block Definition.
- 4 If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar



Command line: BPARAMETER

Use Actions with Dynamic Blocks

Actions define how the geometry of a dynamic block reference will move or change when the custom properties of the block reference are manipulated in a drawing.

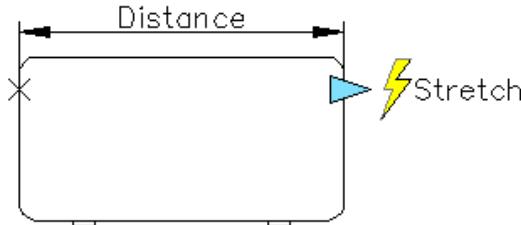
Overview of Using Actions in Dynamic Blocks

Actions define how the geometry of a dynamic block reference will move or change when the custom properties of a block reference are manipulated in a drawing. A dynamic block usually contains at least one action.

In general, when you add an action to a dynamic block definition, you must associate the action with a parameter, a key point on the parameter, and geometry. A *key point* is the point on a parameter that drives its associated action when edited. The geometry associated with an action is called the *selection set*.

In the following example, the dynamic block definition contains geometry that represents a desk, a linear parameter with one grip specified for its

endpoint, and a stretch action associated with the endpoint of the parameter and the geometry for the right side of the desk. The endpoint of the parameter is the key point. The geometry on the right side of the desk is the selection set.



When you want to change the block reference in a drawing, you move the grip, and the desk stretches.



The types of actions you can use in a dynamic block are

- Move
- Scale
- Stretch
- Polar Stretch
- Rotate
- Flip
- Array
- Lookup

You can assign more than one action to the same parameter and geometry. However, you should not assign two or more of the same type of actions to the same key point on a parameter if both actions affect the same geometry. This can result in unexpected behavior in the block reference.

Actions and Parameter Types

Each type of action can be associated with specific parameters. The following table shows the parameters with which you can associate each type of action.

Action Type	Parameter
Move	Point, linear, polar, XY
Scale	Linear, polar, XY
Stretch	Point, linear, polar, XY
Polar Stretch	Polar
Rotate	Rotation
Flip	Flip
Array	Linear, polar, XY
Lookup	Lookup

Use a Move Action in a Dynamic Block

The move action is similar to the MOVE command. In a dynamic block reference, a move action causes objects to move a specified distance and angle.

In a dynamic block definition, you can associate a move action with any of the following parameters:

- Point
- Linear
- Polar
- XY

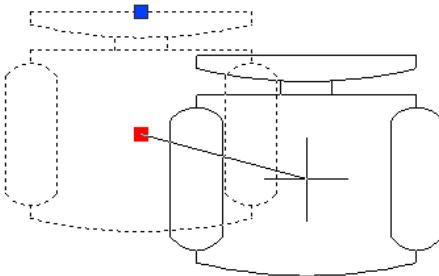
After associating a move action with a parameter, you associate the action with a selection set of geometry.

In a dynamic block reference, if you change the value of a parameter associated with a move action through a grip or the Properties palette, it may affect the key point on the parameter associated with the move action. If the key point is affected, the geometry in the move action's selection set will move.

For example, you have a dynamic block that represents a chair. The block contains a point parameter and a move action associated with the point parameter. The move action's selection set contains all the geometry in the block (the chair). When you use the grip associated with the point parameter

(or the Position X or Position Y properties in the Properties palette) to manipulate the dynamic block reference, it changes the value of the point parameter. This change in value causes the chair to move.

In the following example, when the chair block reference is moved by dragging the point parameter grip, the new position of this grip is reported in the Properties palette.



Specify the Distance Type Property for a Move Action

When a move action is associated with an XY parameter, the move action has an override property called *Distance Type*. This property specifies whether the distance applied to the move is the parameter's X value, Y value, or X and Y coordinate value from the parameter's base point.

For example, you specify X Distance as the Distance Type for a move action in a dynamic block definition. That means that the block can only be moved on the X axis. So if you try to move the block on the Y axis, it doesn't move.

See also:

[Use Distance Multiplier and Angle Offset Action Overrides](#)

To add a move action to a dynamic block definition

- 1 In the Block Editor, in the Block Authoring Palettes window, Actions tab, click the Move Action tool.
- 2 In the Block Editor drawing area, select a parameter to associate with the action. (You can associate the following parameter types with a move action: point, linear, polar, and XY.)
- 3 Select the objects (the selection set) to associate with the action.
- 4 Press ENTER.
- 5 (Optional) Follow the prompts on the command line to specify a distance multiplier and angle offset.

- 6 Do one of the following to specify the location of the action:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.

NOTE The action location in the block definition has no effect on the appearance or functionality of the block reference.

- 7 On the Block Editor toolbar, click Save Block Definition.
- 8 If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar



Command line: BACTION

To specify the Distance Type property for a move action associated with an XY parameter

- 1 In the Block Editor, select a move action.
- 2 On the Properties palette, Overrides area, Distance Type, select an option from the list.

Use a Scale Action in a Dynamic Block

The scale action is similar to the SCALE command. In a dynamic block reference, a scale action causes the block's selection set to scale when the associated parameter is edited by moving grips or by using the Properties palette.

In a dynamic block definition, you associate a scale action with an entire parameter, not a key point on the parameter. You can associate a scale action with any of the following parameters:

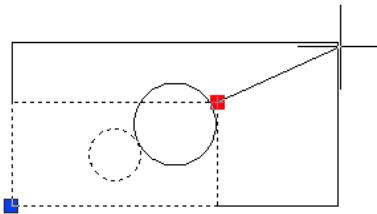
- Linear
- Polar
- XY

After associating a scale action with a parameter, you associate the action with a selection set of geometry.

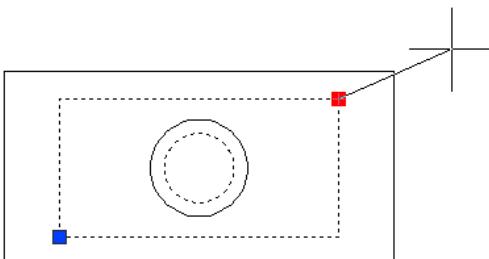
Specify the Type of Base Point for a Scale Action

The scale action has a property called *Base Type*. With this property, you specify whether the base point for the scale factor is *dependent* or *independent*.

If the base type is dependent, the objects in the selection set scale relative to the base point of the parameter with which the scale action is associated. In the following example, a scale action is associated with an XY parameter. The scale action base type is dependent. The base point of the XY parameter is located at the lower left corner of the rectangle. When the custom grip is used to scale the block, it scales relative to the lower-left corner of the rectangle.



If the base type is independent (shown in the Block Editor as an X marker), you specify a base point independent of the parameter with which the scale action is associated. The objects in the selection set will scale relative to this independent base point you specify. In the following example, a scale action is associated with an XY parameter. The scale action base type is independent. The independent base point is located at the center of the circle. When the custom grip is used to scale the block, it scales relative to the center of the circle.



Specify the Scale Type Property for a Scale Action

When a scale action is associated with an XY parameter, the scale action has an override property called *Scale Type*. This property specifies whether the scale factor applied is the parameter's *X* distance, *Y* distance, or *X* and *Y* coordinate value distance from the parameter's base point.

For example, you specify *X Distance* as the *Scale Type* for a scale action in a dynamic block definition. In a drawing, when you edit the block reference by

dragging a grip on the XY parameter only along the Y axis, the associated geometry does not scale.

To add a scale action to a dynamic block definition

- 1 In the Block Editor, in the Block Authoring Palettes window, Actions tab, click the Scale Action tool.
- 2 In the Block Editor drawing area, select a parameter to associate with the action. (You can associate the following parameter types with a scale action: linear, polar, and XY.)
- 3 Select the geometry (the selection set) to associate with the action.
- 4 Press ENTER.
- 5 (Optional) Follow the prompts on the command line to specify the type of base point.
- 6 Do one of the following to specify the location of the action:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.

NOTE The action location in the block definition has no effect on the appearance or functionality of the block reference.

- 7 On the Block Editor toolbar, click Save Block Definition.
- 8 If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar



Command line: BACTION

To specify an independent base point for a scale action in a dynamic block definition

- 1 In the Block Editor, select a scale action.
- 2 On the Properties palette, Overrides area, Base Type, select Independent from the drop-down list.

To specify the Scale Type property for a scale action associated with an XY parameter

- 1 In the Block Editor, select a scale action associated with an XY parameter.
- 2 On the Properties palette, Overrides area, Scale Type, select an option from the drop-down list.

Depending on the override you apply, the scale action for the block is limited to the X axis, the Y axis, or both.

Use a Stretch Action in a Dynamic Block

In a dynamic block reference, a stretch action causes objects to move and stretch a specified distance in a specified location.

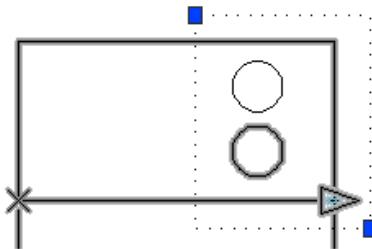
In a dynamic block definition, you can associate a stretch action with any of the following parameters:

- Point
- Linear
- Polar
- XY

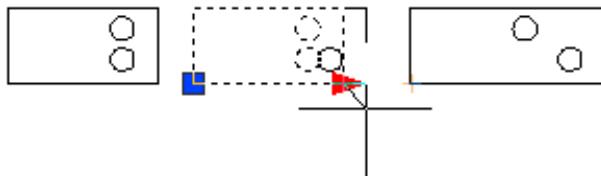
After associating a stretch action with a parameter, you specify a stretch frame for the stretch action. You then select the objects for the stretch action's selection set. The stretch frame determines how the objects within or crossed by the frame are edited in the block reference. The behavior is similar to specifying a crossing selection window with the STRETCH command.

- Objects entirely within the frame are moved.
- Objects that cross the frame are stretched.
- Objects within or crossed by the frame, but not included in the selection set, are not stretched or moved.
- Objects outside the frame and included in the selection set are moved.

In the following example, the stretch frame is indicated by the dashed line, and the selection set has a haloed effect. The top circle, while enclosed by the stretch frame, is not included in the selection set, so the top circle won't move. The bottom circle is entirely enclosed in the stretch frame and included in the selection set, so the bottom circle will move. The rectangle is crossed by the stretch frame and included in the selection set, so the rectangle will stretch.



In a dynamic block reference, if you change the value of a parameter associated with a stretch action through a grip or the Properties palette, it may affect the key point on the parameter associated with the stretch action. If the key point is affected, the geometry in the stretch action's selection set will move.



Specify the Distance Type Property for a Stretch Action

When a stretch action is associated with an XY parameter, the stretch action has an override property called *Distance Type*. This property specifies whether the distance applied to the move is the parameter's X value, Y value, or X and Y coordinate value from the parameter's base point.

For example, you specify X Distance as the Distance Type for a stretch action in a dynamic block definition. In a drawing, when you try to edit the block reference by dragging the key point only along the Y axis, the associated geometry does not move because you've added the Distance Type override allowing only a move on the X axis.

See also:

[Use Distance Multiplier and Angle Offset Action Overrides](#)

To add a stretch action to a block definition

- 1 In the Block Editor, in the Block Authoring Palettes window, Actions tab, click the Stretch Action tool.
- 2 In the Block Editor drawing area, select a parameter to associate with the action. (You can associate the following parameter types with a move action: point, linear, polar, and XY.)

If you associated the stretch action with a point parameter, go to step 4.

- 3 Do one of the following to select a parameter point to associate with the action:
 - Use the pointing device.
 - Follow the prompts on the command line.
- 4 Do one of the following to specify the first corner of the stretch frame:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.
- 5 Do one of the following to specify the opposite corner of the stretch frame:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.
- 6 Select the objects for the selection set.
- 7 Press ENTER.
- 8 (Optional) Follow the prompts on the command line to specify a distance multiplier and angle offset.
- 9 Do one of the following to specify the location of the action:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.

NOTE The action location in the block definition has no effect on the appearance or the functionality of the block reference.

- 10 On the Block Editor toolbar, click Save Block Definition.
- 11 If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar



Command line: BACTION

To specify the Distance Type property for a stretch action associated with an XY parameter

- 1 In the Block Editor, select a stretch action associated with an XY parameter.
- 2 On the Properties palette, Overrides area, Distance Type, select an option from the drop-down list.

Depending on the override you apply, the stretch action for the block is limited to the X axis, the Y axis, or both.

Use a Polar Stretch Action in a Dynamic Block

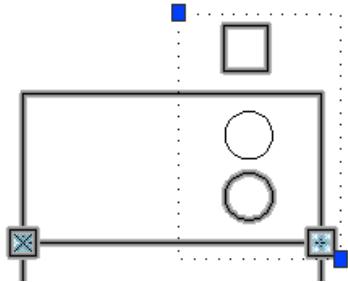
In a block reference, a polar stretch action rotates, moves, and stretches objects a specified angle and distance when the key point on the associated polar parameter is changed through a grip or the Properties palette.

In a dynamic block definition, you can only associate a polar stretch action with a polar parameter. The base point for the stretch part of the action is the parameter point opposite the key point.

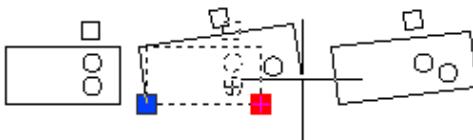
After associating a polar stretch action with a polar parameter, you specify a stretch frame for the polar stretch action. You then select objects to stretch and objects to rotate.

- Objects entirely within the frame are moved.
- Objects that cross the frame are stretched.
- Objects in the action's selection set specified to rotate only are not stretched.
- Objects within the frame are moved linearly after they are rotated.
- Objects crossed by the frame are stretched linearly after they are rotated.
- Objects within or crossed by the frame, but not included in the selection set, are not stretched or rotated.
- Objects outside the frame and included in the selection set are moved.

In the following example, the stretch frame is indicated by the dashed line, and the selection set has a haloed effect. The top circle, while enclosed by the stretch frame, is not included in the selection set, so the top circle won't move. The bottom circle is entirely enclosed in the stretch frame and included in the stretch selection set, so the bottom circle will move. The rectangle is crossed by the stretch frame and included in the selection set, so the rectangle will stretch. The square is entirely enclosed in the stretch frame and included in the rotate selection set, but not the stretch selection set, so the square will rotate only.



In a dynamic block reference, if you change the value of a parameter associated with a polar stretch action through a grip or the Properties palette, it may affect the key point on the parameter associated with the polar stretch action. If the key point is affected, the geometry in the polar stretch action's selection set will move or rotate depending on how the block was defined.



If you only want objects in the block reference to rotate, don't include any objects in the stretch frame.

See also:

[Use Distance Multiplier and Angle Offset Action Overrides](#)

To add a polar stretch action to a block definition

- 1 In the Block Editor, in the Block Authoring Palettes window, Actions tab, click the Stretch Action tool.
- 2 In the Block Editor drawing area, select a polar parameter to associate with the action. (You can associate only a polar parameter with a polar stretch action.)
- 3 Do one of the following to select a parameter point to associate with the action:
 - Use the pointing device.
 - Follow the prompts on the command line.

- 4** Do one of the following to specify the first corner of the stretch frame:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.
- 5** Do one of the following to specify the opposite corner of the stretch frame:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.
- 6** Select the objects to stretch or move.
- 7** Press ENTER.
- 8** Select the objects that you want to rotate.
(The objects you select in steps 6 and 8 make up the polar stretch action's selection set.)
- 9** Press ENTER.
- 10** (Optional) Follow the prompts on the command line to specify distance and offset.
- 11** Do one of the following to specify the location of the action:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.

NOTE The action location in the block definition has no effect on the appearance or functionality of the block reference.

- 12** On the Block Editor toolbar, click Save Block Definition.
- 13** If you are finished using the Block Editor, click Close Block Editor.

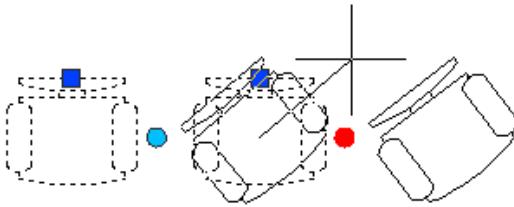
Block Editor toolbar



Command line: BACTION

Use a Rotate Action in a Dynamic Block

The rotate action is similar to the ROTATE command. In a dynamic block reference, a rotate action causes its associated objects to rotate when the associated parameter is edited through a grip or the Properties palette.



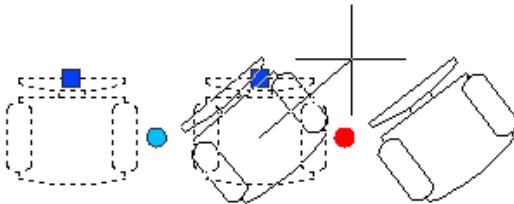
In a dynamic block definition, you can associate a rotate action only with a rotation parameter. The rotate action is associated with the entire parameter, not a key point on the parameter.

After associating a rotate action with a rotation parameter, you associate the action with a selection set of geometry.

Specify the Type of Base Point for a Rotate Action

The rotate action has a property called *Base Type*. With this property, you specify whether the base point for the rotation is the base point of the parameter or an independent base point that you specify in the block definition.

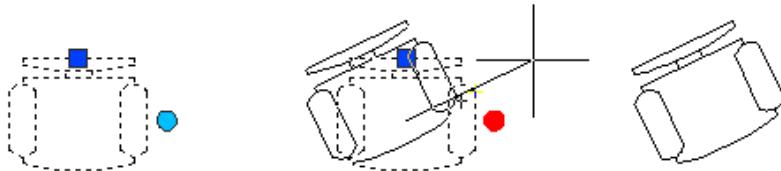
By default, *Base Type* is set to *Dependent*. This means that the block rotates around the associated rotation parameter's base point. In the following example, the chair block contains a rotation parameter and an associated rotate action. The rotate action's *Base Type* is *Dependent*. The base point of the parameter is at the center of the chair. Thus, the chair rotates about the center point.



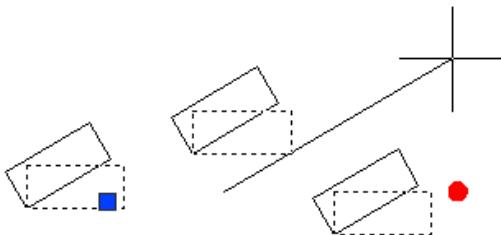
When you set *Base Type* to *Independent*, you can specify a base point for the rotate action other than the base point of the associated rotation parameter. This independent base point is shown in the Block Editor as an X marker. The location of the independent base point can be changed by dragging it or by editing the *Base X* and *Base Y* values in the *Overrides* section of the Properties palette.

In the following example, the chair block contains a rotation parameter and an associated rotate action. The rotate action's *Base Type* is *Independent*. The

independent base point is located at the lower left corner of the chair. Thus, the chair rotates about the lower left corner.



In the following example, each of the three rectangles in the dynamic block reference rotates about an independent base point located at the lower left corner of each rectangle. To achieve this, you assign one rotation parameter. You then add three rotate actions. Each rotate action is associated with the rotate parameter. Each rotate action is then associated with a different object and assigned a different independent base point.



You could achieve the same result by using dependent base points, each with a different base offset, for each rotation action. However, if you need to move the rectangles independently from one another (for example, with a polar or XY parameter and a move action) in the block reference, you should use independent base points for the rotate actions, or the objects won't rotate correctly.

To add a rotate action to a block definition

- 1 In the Block Editor, in the Block Authoring Palettes window, Actions tab, click the Rotate Action tool.
- 2 In the Block Editor drawing area, select a rotation parameter to associate with the action. (You can associate only a rotation parameter with a rotate action.)
- 3 Select the objects (the selection set) to associate with the action.

- 4 Press ENTER.
- 5 (Optional) Follow the prompts on the command line to specify the type of base point.
- 6 Do one of the following to specify the location of the action:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.

NOTE The action location in the block definition has no effect on the appearance or functionality of the block reference.

- 7 On the Block Editor toolbar, click Save Block Definition.
- 8 If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar

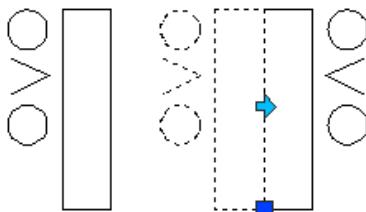


Command line: BACTION

Use a Flip Action in a Dynamic Block

With a flip action, you can flip a dynamic block reference about a specified axis called a reflection line.

In a dynamic block reference, a flip action causes its associated selection set to flip about an axis called a *reflection line* when the associated parameter is edited through a grip or the Properties palette.



In a dynamic block definition, you can associate a flip action only with a flip parameter. The flip action is associated with the entire parameter, not a key point on the parameter. After associating a flip action with a flip parameter, you associate the action with a selection set of geometry. Only the selected objects will flip about the reflection line.

For example, in the following illustration, the polyline between the two circles is not included in the flip action's selection set. When the block reference is flipped, the polyline does not flip with the rest of the geometry.



To add a flip action to a block definition

- 1 In the Block Editor, in the Block Authoring Palettes window, Actions tab, click the Flip Action tool.
- 2 In the Block Editor drawing area, select a parameter to associate with the action. (You can associate only a flip parameter with a flip action.)
- 3 Select the objects (the selection set) to associate with the action.
- 4 Press ENTER.
- 5 Do one of the following to specify the location of the action:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.

NOTE The action location in the block definition has no effect on the appearance or functionality of the block reference.

- 6 On the Block Editor toolbar, click Save Block Definition.
- 7 If you are finished using the Block Editor, click Close Block Editor.

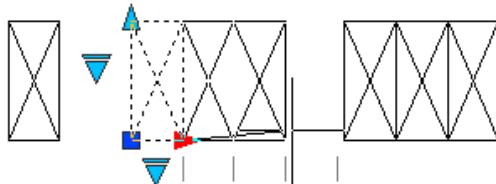
Block Editor toolbar



Command line: BACTION

Use an Array Action in a Dynamic Block

In a dynamic block reference, an array action causes its associated objects to copy and array in a rectangular pattern when the associated parameter is edited through a grip or the Properties palette.



In a dynamic block definition, you can associate an array action with any of the following parameters:

- Linear
- Polar
- XY

After associating an array action with a parameter, you associate the action with a selection set of geometry.

Specify Rows and Columns for an Array Action

When you associate an array action with a linear or polar parameter, you specify the column offset for the arrayed objects. The column offset determines the distance between the arrayed objects. When you edit the parameter in the block reference, the distance of the parameter (from base point to second point) is divided by the column offset to determine the number of columns (the number of objects).

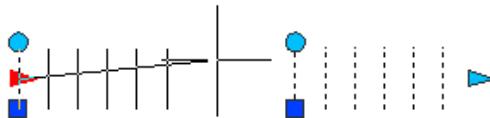
For example, you might associate an array action with a linear parameter. You specify that the array action has a column offset of 2. When you change the linear parameter in the dynamic block reference to a distance of 10, the number of columns for the block reference is 5.

If you associate an array action with an XY parameter, you can also specify the row offset.

Including Parameters in an Array Action's Selection Set

When you include a parameter in an array action's selection set, it has no effect on the behavior of the block reference. The parameter is not copied with the other objects in the selection set. Additional grips are not displayed in the block reference.

In the following example, a parking lot block can be arrayed to contain any number of spaces. The vertical line can also be rotated. Notice that even after the block has been arrayed, it still contains only one rotation grip.



However, when you edit the grip for the parameter included in the array action's selection set, the parameter's associated action is triggered for all instances of the objects. The same behavior occurs when the parameter is not included in the array action's selection set.

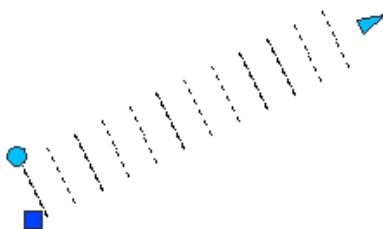
Using Rotate and Array Actions in the Same Dynamic Block

A dynamic block can contain an array action and a rotate action that have the same selection set. The order in which the block reference is arrayed and rotated affects the display of the block.

When you rotate the block first and then array it, all instances of the arrayed objects are individually rotated around their own base point.



When you array the block first and then rotate it, all instances of the arrayed objects are rotated around a single base point.



To add an array action a block definition

- 1 In the Block Editor, in the Block Authoring Palettes window, Actions tab, click the Array Action tool.
- 2 In the Block Editor drawing area, select a parameter to associate with the action. (You can associate linear, polar, and XY parameters with an array action.)

- 3** Select the objects (the selection set) to associate with the action.
- 4** Press ENTER.
- 5** If you are associating the array action with an XY parameter, do one of the following:
 - Enter the distance between rows.
 - Specify a unit cell by entering two values separated by a comma or by picking two opposite points of a rectangle in the drawing area, and then go to step 7.
- 6** Enter a value for the distance between columns.
- 7** If you are assigning the array action to an XY parameter, enter the distance between rows.
- 8** Do one of the following to specify the location of the action:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.

NOTE The action location in the block definition has no effect on the appearance or functionality of the block reference.
- 9** On the Block Editor toolbar, click Save Block Definition.
- 10** If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar



Command line: BACTION

Use a Lookup Action in a Dynamic Block

When you add a lookup action to a dynamic block definition and associate it with a lookup parameter, it creates a lookup table. You can use a lookup table to assign custom properties and values to a dynamic block.

For information about adding lookup tables to dynamic blocks, see Use Lookup Tables to Assign Data to Dynamic Blocks.

To add a lookup action to a dynamic block definition

- 1 In the Block Editor, in the Block Authoring Palettes window, Actions tab, click the Lookup Action tool.
- 2 In the Block Editor drawing area, select one or more lookup parameters to associate with the action. (You can associate only a lookup action with lookup parameters.)
- 3 Do one of the following to specify the location of the action:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.

NOTE The action location in the block definition has no effect on the appearance or functionality of the block reference.

- 4 In the Property Lookup Table dialog box, complete the table as necessary.
- 5 Click OK.
- 6 On the Block Editor toolbar, click Save Block Definition.
- 7 If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar



Command line: BACTION

Use Distance Multiplier and Angle Offset Action Overrides

Some actions have distance multiplier and angle offset override properties. These properties allow you to specify a factor by which a parameter value is increased or decreased when the parameter is edited in a dynamic block reference.

Action overrides are properties of actions. However, they have no effect on the block reference until the block reference is manipulated in a drawing. There are two types of action overrides: distance multiplier and angle offset.

- **Distance multiplier.** You use the distance multiplier property to change a parameter value by a specified factor. For example, if you set the distance multiplier property to 2 for a stretch action, the associated geometry in the block reference would increase and double the distance of the grip movement.

■ **Angle offset.** You use the angle offset property to increase or decrease the angle of a changed parameter value by a specified amount. For example, if you set the angle offset property of a move action to 90, the block reference would move 90 degrees beyond the angle value of the grip movement.

You can specify these action override properties by following the prompts on the command line when you add an action to a dynamic block definition. You can also specify these properties in the Properties palette when you select an action in the Block Editor.

The following table shows the available override properties for each type of action.

Action Type	Available Override Properties
Move	Distance Multiplier, Angle Offset
Scale	None
Stretch	Distance Multiplier, Angle Offset
Polar Stretch	Distance Multiplier, Angle Offset
Rotate	None
Flip	None
Array	None
Lookup	None

To specify a distance multiplier override for an action

- 1 In the Block Editor, select a move, stretch, polar stretch, or array action.
- 2 On the Properties palette, Overrides area, Distance Multiplier, enter a value.
- 3 Press ENTER.

To specify an angle offset override for an action

- 1 In the Block Editor, select a move, stretch, or polar stretch action.
- 2 On the Properties palette, Overrides area, Angle Offset, enter a value.
- 3 Press ENTER.

Use Parameter Sets

Use the Parameter Sets tab on the Block Authoring palette to add commonly paired parameters and actions to your dynamic block definition.

You add a parameter set to your block the same way you add a parameter. The action included in the parameter set is automatically added to your block definition and associated with the added parameter. You must then associate a selection set (geometry) with each action.

When you first add a parameter set to your dynamic block definition, a yellow alert icon is displayed next to each action. This indicates that you need to associate a selection set with each action. You can double click the yellow alert icon (or use the BACTIONSET command) and follow the prompts on the command line to associate the action with a selection set.

NOTE When you insert a lookup parameter set and double click the yellow alert icon, the Property Lookup Table dialog box is displayed. Lookup actions are associated with the data you add to this table, not a selection set.

The following table lists the parameter sets provided on the Parameter Sets tab of the Block Authoring palette.

Parameter Set	Description
Point Move	Adds a point parameter with one grip and an associated move action to the dynamic block definition.
Linear Move	Adds a linear parameter with one grip and an associated move action to the dynamic block definition.
Linear Stretch	Adds a linear parameter with one grip and an associated stretch action to the dynamic block definition.
Linear Array	Adds a linear parameter with one grip and an associated array action to the dynamic block definition.
Linear Move Pair	Adds a linear parameter with two grips and a move action associated with each grip to the dynamic block definition.
Linear Stretch Pair	Adds a linear parameter with two grips and a stretch action associated with each grip to the dynamic block definition.
Polar Move	Adds a polar parameter with one grip and an associated move action to the dynamic block definition.
Polar Stretch	Adds a polar parameter with one grip and an associated stretch action to the dynamic block definition.

Parameter Set	Description
Polar Array	Adds a polar parameter with one grip and an associated array action to the dynamic block definition.
Polar Move Pair	Adds a polar parameter with two grips and a move action associated with each grip to the dynamic block definition.
Polar Stretch Pair	Adds a polar parameter with two grips and a stretch action associated with each grip to the dynamic block definition.
XY Move	Adds an XY parameter with one grip and an associated move action to the dynamic block definition.
XY Move Pair	Adds an XY parameter with two grips and move action associated with each grip to the dynamic block definition.
XY Move Box Set	Adds an XY parameter with four grips and a move action associated with each grip to the dynamic block definition.
XY Stretch Box Set	Adds an XY parameter with four grips and a stretch action associated with each grip to the dynamic block definition.
XY Array Box Set	Adds an XY parameter with four grips and an array action associated with each grip to the dynamic block definition.
Rotation	Adds a rotation parameter with one grip and an associated rotate action to the dynamic block definition.
Flip	Adds a flip parameter with one grip and an associated flip action to the dynamic block definition.
Visibility	Adds a visibility parameter with one grip. No action is required with a visibility parameter.
Lookup	Adds a lookup parameter with one grip and a lookup action to the dynamic block definition.

See also:

[Use Block Authoring Palettes](#)

To add parameter set to a dynamic block definition

- 1 In the Block Editor, in the Block Authoring Palettes window, on the Parameter Sets tab, click a parameter set.
- 2 Follow the prompts on the command line.
- 3 On the Block Editor toolbar, click Save Block Definition.

- 4 (Optional) If you are finished using the Block Editor, click Close Block Editor.

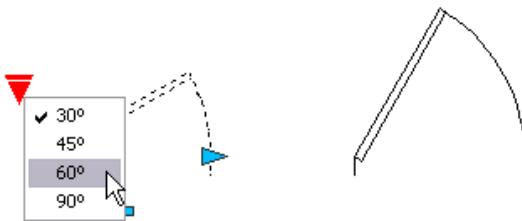
Specify Grips for Dynamic Blocks

When you add a parameter to a dynamic block definition, custom grips associated with key points of the parameter are automatically added to the block. In a drawing, you can use these custom grips to manipulate the geometry of a dynamic block reference.

All parameters (except the alignment parameter, which always displays one grip) have a property called *Number of Grips*. When you select a parameter in the Block Editor, the Number of Grips property is displayed in the Properties palette. This property allows you to specify, from a preset list, the number of grips you want to display for the parameter.

If you specify that a parameter has 0 grips, you can still edit the dynamic block reference through the Properties palette (if the block is defined that way).

If a dynamic block definition contains visibility states or a lookup table, you can define the block so that the only grip that is displayed is a lookup grip. When you click this grip on the block reference, a drop-down list is displayed. When you select an item from the list, the display of the block reference may change.



Grips are automatically added at key points on the parameter. You can reposition a grip anywhere in the block space relative to its associated key point on the parameter. When you reposition a grip, it is still tied to the key point with which it is associated. No matter where the grip is shown in the block reference, it will still manipulate the key point with which it is associated. If you move or change the key point of the parameter, the grip's position relative to the key point is maintained. Because you use grips to manipulate dynamic block references in a drawing, you should make sure that each grip is placed in a logical location. If a grip is not in a logical place, the behavior may not be what is expected.

Both linear and polar parameters can display two, one, or no grips. When you define a linear or polar parameter to display one grip, it is displayed at the

endpoint of the parameter. You should only assign actions to the endpoint of either of these parameters if you plan to display only one grip. Otherwise, you will not be able to manipulate the reference (trigger the actions) because a grip won't be displayed for the key point associated with the action.

If you reposition the grips for a dynamic block, you can use the BGRIPSET command to reset the grips to their default locations.

The type of parameter you add to the dynamic block definition determines the type of grips that are added to the block. These grips give visual clues as to how the block reference can be manipulated in a drawing. The following table details the types of grips used in dynamic blocks, what they look like, and the parameters with which they are associated.

Grip Type	How the Grip Can Be Manipulated in a Drawing	Associated Parameters
Standard		Within a plane in any direction Base, Point, Polar, and XY
Linear		Back and forth in a defined direction or along an axis Linear
Rotation		Around an axis Rotation
Flip		Click to flip the dynamic block reference Flip
Alignment		Within a plane in any direction; when moved over an object, triggers the block reference to align with the object Alignment
Lookup		Click to display a list of items Visibility, Lookup

Specify Insertion Cycling for Grips in Dynamic Blocks

Grips in dynamic blocks have a property called *Cycling*. When this property is set to Yes, the grip becomes an available insertion point for the dynamic block reference. You can use the BCYCLEORDER command to turn on and off cycling for grips in dynamic blocks as well as specify the cycling order for the grips. When you insert the dynamic block reference in a drawing, you can use the CTRL key to cycle through the available grips to select which grip will be the insertion point for the block.

To specify the number of custom grips displayed for a parameter in a dynamic block

- 1 In the Block Editor, select a parameter.
- 2 On the Properties palette, under Misc, click Number of Grips, and then select the number of grips you want to display for the parameter.
- 3 On the Block Editor toolbar, click Save Block Definition.
- 4 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Command line: BGRIPSET

Shortcut menu: In the Block Editor, select a parameter. Right-click in the drawing area. Click Grip Display ► *any value*.

To reposition a grip in a dynamic block definition

- 1 In the Block Editor, select a custom grip (not the standard grip associated with the key point of the parameter).
- 2 Do one of the following to reposition the grip:
 - Drag the grip to another position in the block space.
 - On the Properties palette, under Geometry, enter values for the Base X and Base Y properties. (These values are relative to the base point of the parameter.)
- 3 On the Block Editor toolbar, click Save Block Definition.
- 4 (Optional) If you are finished using the Block Editor, click Close Block Editor.

To reset grips in a dynamic block definition to their default locations

- 1 In the Block Editor, enter **bgriset** on the command line.
- 2 Select the grip for which you want to reset the grip position(s).
- 3 On the command line, enter **reposition**, and then press ENTER.
- 4 On the Block Editor toolbar, click Save Block Definition.
- 5 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Shortcut menu: In the Block Editor, select a parameter. Right-click in the drawing area. Click Grip Display ► Reset Position.

To turn insertion cycling on or off for a custom grip in a dynamic block reference

- 1 In the Block Editor, enter **bcycleorder** on the command line.
- 2 In the Insertion Cycling Order dialog box, select a grip from the list, and click Cycling to turn cycling on or off for the grip. (A check mark in the Cycling column indicates that cycling is turned on for the grip.)
- 3 Click OK.
- 4 On the Block Editor toolbar, click Save Block Definition.
- 5 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Shortcut menu: In the Block Editor, select a dynamic grip. Right-click in the drawing area. Click Insertion Cycling.

To modify the insertion cycling order for custom grips in a dynamic block reference

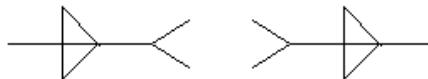
- 1 In the Block Editor, enter **bcycleorder** on the command line.
- 2 In the Insertion Cycling Order dialog box, select a grip from the list, and click Move Up or Move Down. (A check mark in the Cycling column indicates that cycling is turned on for the grip.)
- 3 Repeat step 2 until you are finished modifying the grip cycling order.
- 4 Click OK.
- 5 On the Block Editor toolbar, click Save Block Definition.
- 6 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Create Visibility States

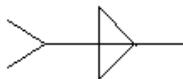
You can use visibility states to make geometry visible or invisible in a dynamic block. One block can have any number of visibility states.

Using visibility states is a powerful way to create a block that has many different graphical representations. You can easily change a block reference that has different visibility states without having to find a different one to insert in your drawing.

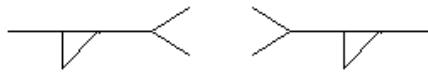
For example, you have the following four different weld symbols.



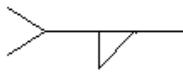
WLD1



WLD2

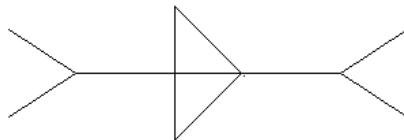


WLD3



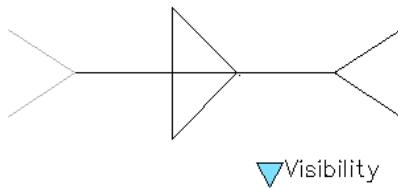
WLD4

Using visibility states, you can combine these weld symbols into a single dynamic block. The following example shows the geometry for the four weld symbols combined in one dynamic block definition in the Block Editor.

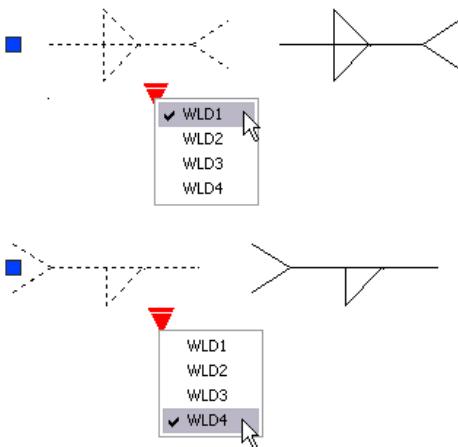


After you combine the geometry in the Block Editor, you add a visibility parameter. You add only one visibility parameter to a dynamic block definition. You do not associate any actions with a visibility parameter.

You can then create and name a different visibility state for each weld symbol (for example, WLD1, WLD2, WLD3, and WLD4). You can make particular geometry visible or invisible for each state. In the following example, the WLD1 visibility state is displayed in the Block Editor. The geometry that displays in a dimmed state is invisible for the WLD1 visibility state.



The visibility parameter includes a lookup grip. This grip is always displayed in a block reference that contains visibility states. When you click the grip in the block reference, a drop-down list of all the visibility states in the block reference is displayed. When you select one of the states from the list, the geometry that is visible for that state is displayed in the drawing.



The right-hand side of the Block Editor toolbar displays the name of the current visibility state. All blocks have at least one visibility state. You cannot delete the current state. This area of the tool bar also provides several tools for working with visibility states.

When you work with visibility states, you may or may not want to see the geometry that is invisible for a given state. You can use the Visibility Mode button (BVMODE) to display or not display geometry (in a dimmed state) that is invisible. When the BVMODE system variable is set to 1, geometry that is invisible for a given state displays in a dimmed mode.

To enable visibility states (add a visibility parameter)

- 1 In the Block Editor, in the Block Authoring Palettes window, Parameters tab, click the Visibility Parameter tool.
- 2 (Optional) Follow the prompts on the command line to specify the following for the parameter:
 - Name
 - Label
 - Description
 - Properties displayed for the block reference

NOTE You can also specify and edit these properties in the Properties palette at a later time, after you've added the parameter to the block definition.

- 3** Do one of the following to specify a location for the parameter:
 - Use the pointing device.
 - Enter the X and Y coordinate values, separated by a comma.

This is the default location for the visibility parameter grip.
- 4** On the Block Editor toolbar, click Save Block Definition.
- 5** (Optional) If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar



Command line: BPARAMETER

To create a new visibility state based on the current state

- 1** On the Block Editor toolbar, click Manage Visibility States.
- 2** In the Visibility States dialog box, click New.
- 3** In the New Visibility State dialog box, enter a name for the new visibility state.
- 4** Click Leave Visibility of Existing Objects Unchanged in New State.
- 5** Click OK.

The visibility of existing objects is unchanged in the new state.

Block Editor toolbar



Command line: BVSTATE

To create a new visibility state in which all objects are invisible

- 1** On the Block Editor toolbar, click Manage Visibility States.
- 2** In the Visibility States dialog box, click New.
- 3** In the New Visibility State dialog box, enter a name for the new visibility state.
- 4** Click Hide All Existing Objects in New State.

5 Click OK.

Block Editor toolbar



Command line: BVSTATE

To create a new visibility state in which all objects are visible

- 1** On the Block Editor toolbar, click Manage Visibility States.
- 2** In the Visibility States dialog box, click New.
- 3** In the New Visibility State dialog box, enter a name for the new visibility state.
- 4** Click Show All Existing Objects in New State.
- 5** Click OK.

Block Editor toolbar



Command line: BVSTATE

To make a visibility state current

- 1** On the Block Editor toolbar, click the Visibility States control.
- 2** Select the visibility state that you want to set as the current state.

Block Editor toolbar



Command line: BVSTATE

Shortcut menu: In the Visibility States dialog box, right-click a visibility state in the list. Click Set Current.

To delete a visibility state

- 1** On the Block Editor toolbar, click Manage Visibility States.
- 2** In the Visibility States dialog box, select a visibility state in the list.

3 Click Delete.

4 Click OK.

Block Editor toolbar



Command line: BVSTATE

Shortcut menu: In the Visibility States dialog box, right-click a visibility state in the list. Click Delete State.

To rename a visibility state

1 In the Block Editor, click Manage Visibility States.

2 In the Visibility States dialog box, select a visibility state in the list.

3 Click Rename.

4 Enter a new name for the visibility state.

5 Click OK.

Block Editor toolbar



Command line: BVSTATE

Shortcut menu: Right-click a visibility state in the list. Click Rename State.

To make objects visible in the current visibility state

1 On the Block Editor toolbar, click Make Visible.

2 In the drawing area, select the objects to make visible in the current visibility state.

3 Press ENTER.

4 On the command line, enter **current**.

5 Press ENTER.

Block Editor toolbar



Command line: BVSHOW

Shortcut menu: In the Block Editor drawing area, select one or more objects. Right-click in the drawing area. Click Object Visibility ► Show for Current State.

To make objects visible in all visibility states

- 1 In the Block Editor drawing area, select the objects that you want to make visible in all visibility states.
- 2 Right-click in the drawing area.
- 3 Click Object Visibility ► Show for All States.

Command line: BVSHOW

To make objects invisible in the current visibility state

- 1 On the Block Editor toolbar, click Make Invisible.
- 2 In the drawing area, select the objects that you want to make invisible in the current visibility state.
- 3 Press ENTER.
- 4 On the command line, enter **current**.
- 5 Press ENTER.

Block Editor toolbar



Command line: BVHIDE

Shortcut menu: In the Block Editor drawing area, select one or more objects. Right-click in the drawing area. Click Object Visibility ► Hide for Current State.

To make objects invisible in all visibility states

- 1 In the drawing area, select the objects that you want to make invisible in all visibility states.

- 2** Right-click in the drawing area.
- 3** Click Object Visibility ► Hide for All States.

Command line: BVHIDE

To display or not display invisible geometry for visibility states

- 1** On the Block Editor toolbar, click Visibility Mode.

Block Editor toolbar



Command line: BVMODE

To change the order of the visibility states list for a dynamic block reference

- 1** On the Block Editor toolbar, click Manage Visibility States.
- 2** In the Visibility States dialog box, select a visibility state in the list.
- 3** Click Move Up or Move Down to change the order.
- 4** Repeat steps 2 and 3 until the list of visibility states is in the correct order.
- 5** Click OK.

Block Editor toolbar



Command line: BVSTATE

To set the default visibility state for the dynamic block reference

- 1** In the Block Editor, click Manage Visibility States.
- 2** In the Visibility States dialog box, select a visibility state in the list.
- 3** Click Move Up until the selected visibility state is at the top of the list.
- 4** Click OK.

Block Editor toolbar



Command line: BVSTATE

Use Lookup Tables to Assign Data to Dynamic Blocks

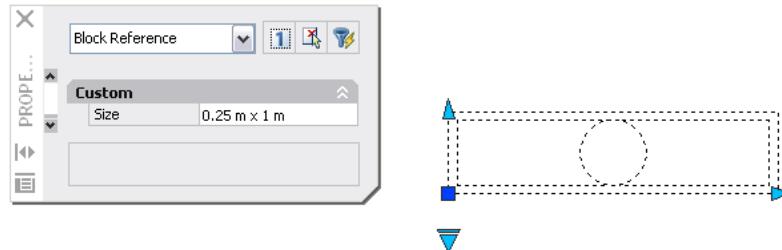
You can use a lookup table to define properties for and assign property values to a dynamic block.

Overview of Using Lookup Tables to Assign Data to Dynamic Blocks

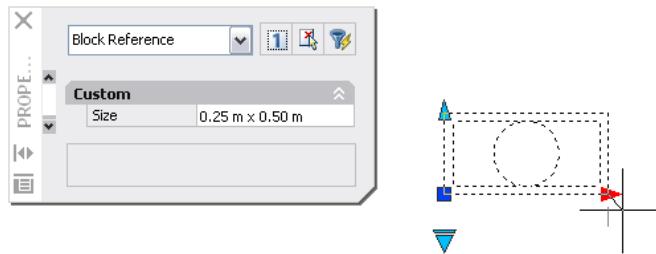
You can use a *lookup table* to define properties for and assign property values to a dynamic block. Using lookup tables is a powerful way to associate parameter values for the dynamic block reference with other data that you specify (for example, a model or part number). You can extract this data from block references in a drawing just as you would extract block attribute data.

When completed, the lookup table assigns property values to the dynamic block reference based on how it is manipulated in a drawing. Conversely, you can change how the block reference is displayed in the drawing by changing the value of a lookup property of the block reference through a lookup grip or the Properties palette.

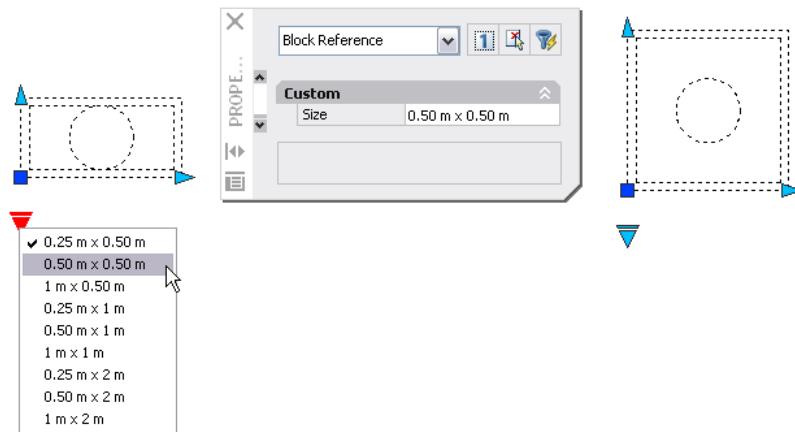
The following example shows a dynamic block reference for a lighting fixture. For this block, the Custom area of the Properties palette displays a size property. This property is defined in the lookup table for the block.



If you change the size of the lighting fixture in your drawing (using the custom grips), the size property in the Properties palette changes accordingly.



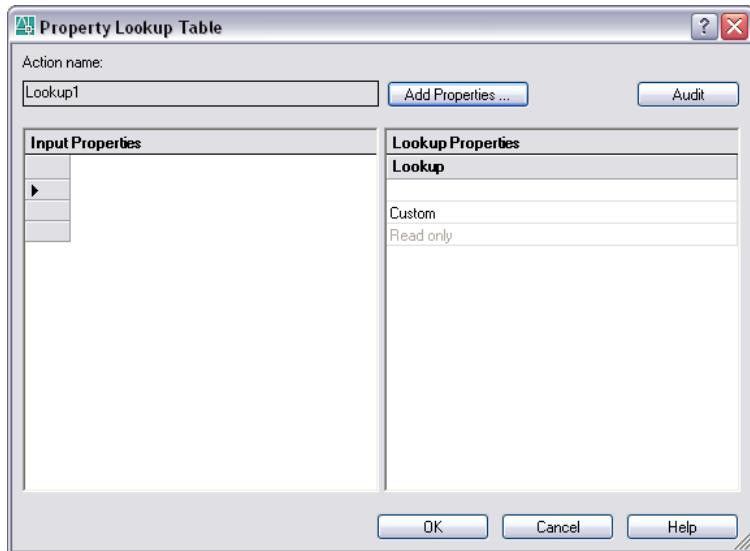
You can also use the lookup grip to change the display of the block reference. When you select a size from the list that is displayed when you click the grip, the block's geometry changes and the new size is displayed in the Properties palette under Custom.



Create a Lookup Table

After you've drawn the geometry and added the appropriate parameters and actions required for the dynamic block's functionality, you can add a lookup table to the block definition.

A blank lookup table is created when you add a lookup parameter to a dynamic block definition and then add a lookup action and associate it with the lookup parameter. The lookup table is displayed in the lookup table dialog box.



Add Properties and Values to a Lookup Table

The Property Lookup Table consists of input properties and lookup properties. Each input property and lookup property is represented by a column in the table.

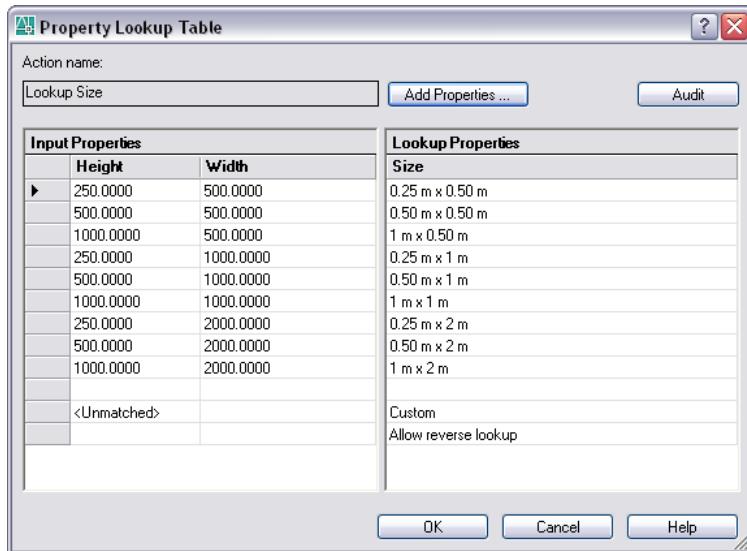
- **Input properties.** Parameters other than lookup parameters (for example, a linear parameter labeled "Width"). You can create a column under Input Properties for every parameter in the block definition except for lookup, alignment, and base point parameters.
- **Lookup properties.** Lookup parameters. You add one lookup parameter to the dynamic block definition for every lookup property column you want to add to the lookup table. The lookup parameter label is used as the property name.

After you add the properties (columns) to the table, you add values to the cells in each column. You click a cell and enter a value. Make sure to follow the guidelines in *Specify Values for Lookup Tables*.

If you defined a value set for an input property (parameter) in the table, a drop-down list of available values will display when you click a cell in that column.

The following lookup table is used for the lighting fixture block shown previously. The linear parameters that have been labeled "Height" and "Width" have been added as input properties. "Size" has been added as a lookup

property. The values under Height and Width are taken from the value sets defined for each of those parameters. The values under Size are entered manually.



When the parameter values in a dynamic block reference match a row of input property values, the corresponding lookup property values in that table row are assigned to the block reference. These lookup properties and values are displayed in the Properties palette under Custom.

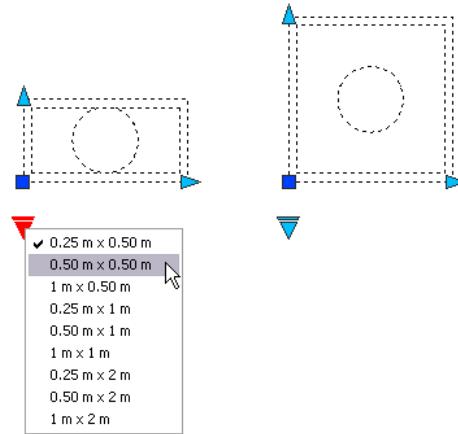
For example, when you edit the lighting fixture block reference to have a height of 250 centimeters and a width of 500 centimeters, the Size property in the Custom area of the Properties palette will display a value of 0.25 m x 0.50 m.

In the lookup table, at the bottom of the input properties is a row labeled <*Unmatched*>. When the parameter values in a block reference do not match a row of input property values defined in the block's lookup table, the lookup property value that corresponds to <*Unmatched*> is assigned to the block reference. The default value is *Custom*, but you can specify another value by changing it in the table.

Enable Reverse Lookup

You can also specify that a lookup property allows for reverse lookup. This adds a lookup grip to the dynamic block reference. When you click this grip, a drop-down list of the lookup values for that lookup property (column in the table) is displayed. When you select a value from the list, the corresponding input property values are assigned to the block reference. Depending on how

the block was defined, this usually results in a change in the block reference's geometry.



To enable reverse lookup for a lookup property, each row in the lookup table must be unique. After you add properties and values to the table, you can check for errors and empty cells by clicking the audit button in the Property Lookup Table dialog box. You can also use the shortcut menu options to insert, delete, or reorder rows.

NOTE If you leave a cell empty in the lookup properties column and the cell has an associated input property in the same row, the lookup property will not allow for reverse lookup.

See also:

- [Extract Data from Block Attributes](#)
- [Extract Block Attribute Data \(Advanced\)](#)

To create a lookup table for a dynamic block

- 1 In the Block Editor, add one or more lookup parameters to a dynamic block definition by following the steps in “To add a lookup parameter to a dynamic block definition” on page 495.
- 2 In the Block Editor, add a lookup action to a dynamic block definition by following the steps in “To add a lookup action to a dynamic block definition” on page 516.

The Property Lookup Table dialog box is displayed.

- 3** Use any of the following procedures to set up a lookup table for the dynamic block:
 - To add input properties to a lookup table
 - To add lookup properties to a lookup table
 - To enable reverse lookup
- 4** On the Block Editor toolbar, click Save Block Definition.
- 5** (Optional) If you are finished using the Block Editor, click Close Block Editor.

Block Editor toolbar



Command line: BPARAMETER

To add input properties to a lookup table

- 1** In the Block Editor, double-click a lookup action in a dynamic block definition.
- 2** In the Property Lookup Table dialog box, click Add Properties.
- 3** In the Add Parameter Properties dialog box, in the bottom-left corner of the dialog box, click Add Input Properties.
- 4** In the Parameter Properties list, select the parameter properties you want to add to the lookup table. Hold CTRL down to select more than one property.
- 5** Click OK.

Command line: BLOOKUPTABLE

To add lookup properties to a lookup table

- 1** In the Block Editor, double-click a lookup action in a dynamic block definition.
- 2** In the Property Lookup Table dialog box, click Add Properties.
- 3** In the Add Parameter Properties dialog box, in the bottom-left corner of the dialog box, click Add Lookup Properties.

- 4 In the Parameter Properties list, select the lookup parameter properties you want to add to the lookup table. Hold CTRL down to select more than one property.
- 5 Click OK.

Command line: BLOOKUPTABLE

To add values to a lookup table

- 1 In the Block Editor, double-click a lookup action in a dynamic block definition.
- 2 In the Property Lookup Table dialog box, click an empty cell and do one of the following:
 - Select a value from the drop-down list. (A drop-down list only displays if a value set has been defined for the parameter.)
 - Enter a value.
- 3 Repeat Step 2 as required to complete the table.
- 4 Click OK.

Command line: BLOOKUPTABLE

To enable reverse lookup

- 1 In the Block Editor, double-click a lookup action in a dynamic block definition.
- 2 In the Property Lookup Table dialog box, at the bottom of a Lookup Property column, click Read Only, and then select Allow Reverse Lookup from the list.

NOTE You can only select Allow Reverse Lookup if you have entered data in the column.

- 3 Click OK.

Command line: BLOOKUPTABLE

To make the lookup property read-only

- 1 In the Block Editor, double-click a lookup action in a dynamic block definition.

- 2 In the Property Lookup Table dialog box, at the bottom of a Lookup Property column, click Allow Reverse Lookup, and then select Read Only from the list.
- 3 Click OK.

Command line: BLOOKUPTABLE

To check for errors in a lookup table

- 1 In the Property Lookup Table dialog box, click Audit.

To open an existing lookup table

- 1 In the Block Editor, double-click a lookup action.

Command line: BLOOKUPTABLE

Specify Values for Lookup Tables

The following rules apply when you specify values in lookup tables:

- Use a comma as the delimiter between values.
- You can specify any number of unique values separated by commas. For example: 5,6,7 5.5,6.25
- To specify a range, use brackets [] to specify that the range includes the values separated by a comma, or use parentheses () to specify that the range does not include the values separated by a comma.
- For a continuous range, use a pair of values separated by a comma, enclosed in brackets or parentheses. For example: [3,10] specifies any value between 3 and 10, including 3 and 10 (3,10) specifies any value between 3 and 10, not including 3 and 10.
- For an open-ended range, use one value with a comma, enclosed in brackets or parentheses. For example: [,5] specifies less than or equal to 5; (5,) specifies greater than 5.
- Use no more than 256 characters in a table cell.
- You can use architectural and mechanical unit syntax (for example, 15'1/4").
- If you enter a value in an invalid format, the value will be reset to the last value when you move to another cell in the table.

Lookup tables support the following:

- All numeric parameter properties (for example, distance and angles for point, linear, polar, XY, and rotation parameters)
- Text string parameter properties (for example, flip and visibility parameter values)

No procedures for this topic.

Specify Custom Properties for Dynamic Blocks

When you create a dynamic block, you can specify custom properties for the block. You can also specify whether or not these properties appear in the Properties palette when you select the block reference in a drawing.

Overview of Specifying Custom Properties for Dynamic Blocks

In the Block Editor, you can specify properties for a parameter in a dynamic block definition. Some of these properties can be displayed as *custom properties* for the dynamic block reference when it is in a drawing. These properties are displayed under Custom in the Properties palette.

In the Block Editor, you can specify parameter labels. When you select the dynamic block reference in a drawing, these properties are shown under Custom in the Properties palette. It is good practice to specify unique parameter labels within the block.

Depending on the parameters used in the dynamic block definition, other parameter properties may be listed under Custom in the Properties palette when you select the dynamic block reference in a drawing. For example, a polar parameter has an angle property that displays in the Properties palette. Depending on how the dynamic block is defined, these properties might display values such as size, angle, and position for a selected block reference.

You can specify whether or not these custom properties are displayed for the block reference when it is selected in a drawing. These properties can also be extracted using the Attribute Extraction wizard.

You can also use the Properties palette to specify the geometric properties (such as color, linetype, and linewidth) of a parameter. These properties are listed in the Properties Palette under Geometry when you select a parameter in the Block Editor.

Other parameter properties, such as Value Set properties and Chain Actions, define how the block reference will function in a drawing.

In the Block Editor, you can also specify if the block can be exploded and if the block can be non-uniformly scaled.

See also:

[“Specify Value Sets for Dynamic Blocks” on page 542](#)

[Allow Chained Actions for Dynamic Blocks](#)

[Specify Grips for Dynamic Blocks](#)

[Extract Data from Block Attributes](#)

To specify a parameter label

- 1 In the Block Editor, click Tools menu ► Properties to display the Properties palette.
- 2 In the Block Editor, select a parameter.
- 3 In the Properties palette, under Property Labels, click *<parameter name>* label.
- 4 Enter a label for the parameter.
- 5 On the Block Editor toolbar, click Save Block Definition.
- 6 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar



Command line: PROPERTIES

Shortcut menu: Right-click the selected object. Click Rename Label. Enter a new name for the parameter, and press ENTER.

To specify the display of custom properties in a dynamic block reference

- 1 In the Block Editor, click Tools menu ► Properties to display the Properties palette.
- 2 In the Block Editor, select a parameter.
- 3 In the Properties palette, under Misc, click Show Properties.
- 4 On the drop-down list, do one of the following:
 - Select Yes to display custom properties for the block reference.
 - Select No to specify that the custom properties will not display for a block reference.

- 5 On the Block Editor toolbar, click Save Block Definition.
- 6 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar



Command line: PROPERTIES

To specify exploding dynamic block reference

- 1 Use one of the following procedures to open a block definition in the Block Editor:
 - To open an existing block definition in the Block Editor
 - To open a block from the DesignCenter window in the Block Editor
- 2 In the Block Editor, make sure nothing is selected.
- 3 In the Properties palette, under Block, click Allow Exploding.
- 4 On the drop-down list, select Yes or No.
- 5 On the Block Editor toolbar, click Save Block Definition.
- 6 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar



Command line: BEDIT

To prevent non-uniform scaling in dynamic block reference

- 1 Use one of the following procedures to open a block definition in the Block Editor:
 - To open an existing block definition in the Block Editor
 - To open a block from the DesignCenter window in the Block Editor
- 2 In the Block Editor, make sure nothing is selected.
- 3 In the Properties palette, under Block, click Scale Uniformly.

- 4 On the drop-down list, select Yes.
- 5 On the Block Editor toolbar, click Save Block Definition.
- 6 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar



Command line: BEDIT

Specify Value Sets for Dynamic Blocks

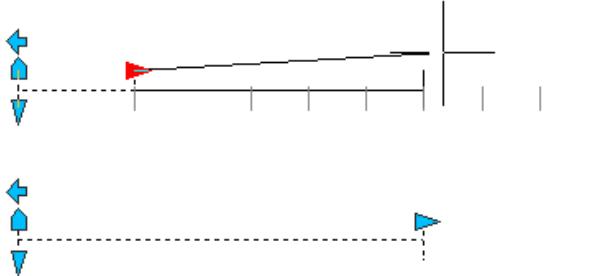
You can specify a defined *value set* for linear, polar, XY, and rotation parameters.

A value set is a range or list of values specified for a parameter. These values can be displayed for the block reference as a drop-down list next to the parameter label under Custom in the Properties palette. When you define a value set for a parameter, the parameter is limited to these values when the block reference is manipulated in a drawing. For example, if you define a linear parameter in a block that represents a window to have a value set of 20, 40, and 60, the window can only be stretched to 20, 40, or 60 units.

When you create a value list for a parameter, the value of the parameter as it exists in the definition is automatically added to the value set. This is the default value for the block reference when you insert it in a drawing.

In a block reference, if you change the parameter's value to a value other than one in the list, the parameter will adjust to the closest valid value. For example, you define a linear parameter to have a value set of 2, 4, and 6. When you try to change the value of that parameter in a block reference to 10, the resulting parameter value is 6 because it is the closest valid value.

When you specify a value set for a parameter in a dynamic block, tick marks are displayed when you grip-edit the block reference in a drawing. The tick marks indicate the locations for the parameter's valid values.



NOTE If you redefine the values in a value set after you've added the parameter properties to a lookup table, make sure to update the lookup table to match the new values in the value set.

To specify a value set for a linear or polar parameter

- 1 In the Block Editor, click Tools menu > Properties to display the Properties palette.
- 2 In the Block Editor, select a linear or polar parameter.
- 3 In the Properties palette, under Value Set, click Dist Type.
- 4 On the drop-down list, select List.
- 5 In the Properties palette, under Value Set, click Dist Value List.
- 6 Click the [...] button.
- 7 In the Add Distance Value dialog box, under Distances to Add, enter one value or two or more values separated by commas.
- 8 Click Add.
- 9 Click OK.
- 10 On the Block Editor toolbar, click Save Block Definition.
- 11 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar



Command line: PROPERTIES

To specify a value set for an XY parameter

- 1 In the Block Editor, click Tools menu ► Properties to display the Properties palette.
- 2 In the Block Editor, select an XY parameter.
- 3 In the Properties palette, under Value Set, click Hor Type or Ver Type.
- 4 On the drop-down list, select List.
- 5 In the Properties palette, under Value Set, click Hor Value List or Ver Value List.
- 6 Click the [...] button.
- 7 In the Add Distance Value dialog box, under Distances to Add, enter one value or two or more values separated by commas.
- 8 Click Add.
- 9 Click OK.
- 10 On the Block Editor toolbar, click Save Block Definition.
- 11 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar



Command line: PROPERTIES

To specify a value set for a rotation parameter

- 1 In the Block Editor, click Tools menu ► Properties to display the Properties palette.
- 2 In the Block Editor, select a rotation parameter.
- 3 In the Properties palette, under Value Set, click Ang Type.
- 4 On the drop-down list, select List.
- 5 In the Properties palette, under Value Set, click Ang Value List.
- 6 Click the [...] button.
- 7 In the Add Angle Value dialog box, under Angle to Add, enter one value or two or more values separated by commas.
- 8 Click Add.

- 9** Click OK.
- 10** On the Block Editor toolbar, click Save Block Definition.
- 11** (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar



Command line: PROPERTIES

To delete values from a parameter's value set

- 1** In the Block Editor, click Tools menu ► Properties to display the Properties palette.
- 2** In the Block Editor, select a linear, polar, XY, or rotation parameter.
- 3** In the Properties palette, under Value Set, click one of the following:
 - Dist Value List
 - Ang Value List
 - Hor Value List
 - Ver Value List
- 4** Click the [...] button.
- 5** In the Add Distance Value or Add Angle Value dialog box, select a value from the list to delete.
- 6** Click Delete.
- 7** Click OK.
- 8** On the Block Editor toolbar, click Save Block Definition.
- 9** (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar



Command line: PROPERTIES

To specify an incremental value set for a linear, polar, XY, or rotation parameter

- 1 In the Block Editor, click Tools menu ► Properties to display the Properties palette.
- 2 In the Block Editor, select a linear, polar, XY, or rotation parameter.
- 3 In the Properties palette, under Value Set, click Dist Type, Ang Type, Hor Type, or Ver Type.
- 4 On the drop-down list, select Increment.
- 5 In the Properties palette, under Value Set, click Dist Increment, Ang Increment, Hor Increment, or Ver Increment, and then enter an incremental value for the parameter.
- 6 Click Dist Minimum, Ang Minimum, Hor Minimum, or Ver Minimum, and then enter a minimum value for the parameter.
- 7 Click Dist Maximum, Ang Maximum, Hor Maximum, or Ver Maximum, and then enter a maximum value for the parameter.
- 8 On the Block Editor toolbar, click Save Block Definition.
- 9 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar



Command line: PROPERTIES

To specify minimum and maximum values for a linear, polar, XY, or rotation parameter

- 1 In the Block Editor, click Tools menu ► Properties to display the Properties palette.
- 2 In the Block Editor, select a linear, polar, XY, or rotation parameter.
- 3 In the Properties palette, under Value Set, click Dist Minimum, Ang Minimum, Hor Minimum, or Ver Minimum, and then enter a minimum value for the parameter.
- 4 Click Dist Maximum, Ang Maximum, Hor Maximum, or Ver Maximum, and then enter a maximum value for the parameter.
- 5 On the Block Editor toolbar, click Save Block Definition.

- 6** (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar



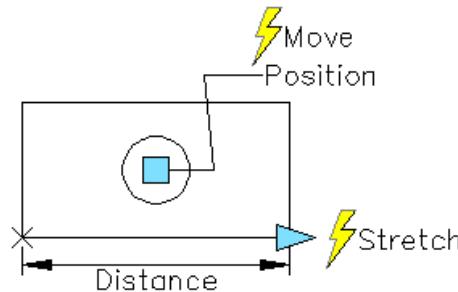
Command line: PROPERTIES

Allow Chained Actions for Dynamic Blocks

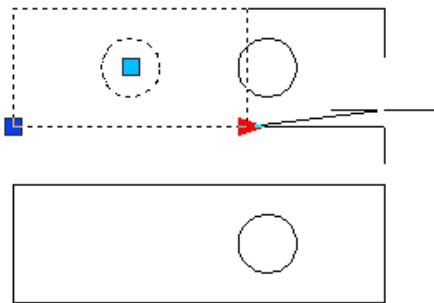
Point, linear, polar, XY, and rotation parameters have a property called *Chain Actions*. This property affects the parameter behavior if the parameter is part of an action's selection set.

For example, you might include a point parameter in the selection set of a stretch action that is associated with a linear parameter. When the linear parameter is edited in a block reference, its associated stretch action triggers a change in its selection set. Because the point parameter is included in the selection set, the point parameter is edited by the change in the linear parameter.

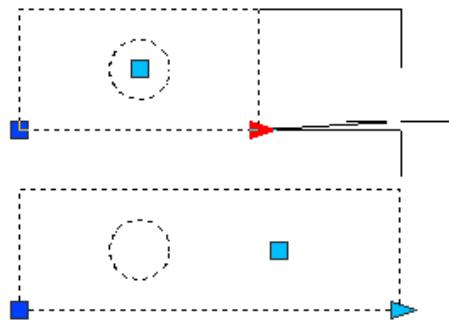
The following example shows a block definition in the Block Editor. The point parameter (labeled Position) is included in the stretch action's selection set.



If the Chain Actions property for the point parameter is set to Yes, a change in the linear parameter will trigger the move action associated with the point parameter, just as if you edited the point parameter in the block reference through a grip or custom property.



If the Chain Actions property is set to No, the point parameter's associated move action is not triggered by the changes to the linear parameter. Thus, the circle doesn't move.



To specify the Chain Actions property for a point, linear, polar, XY, or rotation parameter

- 1 In the Block Editor, click Tools menu > Properties to display the Properties palette.
- 2 In the Block Editor, select a point, linear, polar, XY, or rotation parameter.
- 3 In the Properties palette, under Misc, click Chain Actions.
- 4 On the drop-down list, select Yes or No.
- 5 On the Block Editor toolbar, click Save Block Definition.
- 6 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar



Command line: PROPERTIES

Modify Dynamic Block Definitions

After you've defined a dynamic block, you can modify it in the Block Editor. You can delete, add, and modify the following elements in the Block Editor:

- Parameters (see Use Parameters in Dynamic Blocks)
- Actions (see Use Parameters in Dynamic Blocks Use Actions with Dynamic Blocks)
- Geometry
- Visibility states (see Create Visibility States)
- Lookup tables (see Use Lookup Tables to Assign Data to Dynamic Blocks)
- Grips (see Specify Grips for Dynamic Blocks)
- Properties (see Overview of Specifying Custom Properties for Dynamic Blocks)

You can also use the BACTIONSET command to change an action's selection set. If you delete a parameter associated with an action and need to reassign that action to another parameter, use the BASSOCIATE command.

After you modify a dynamic block definition in the Block Editor, you must save your changes (see Save a Block in the Block Editor).

WARNING If you redefine a dynamic block in AutoCAD 2005 or earlier, the block will lose its dynamic behavior.

To open an existing dynamic block definition in the Block Editor

- 1 Open the drawing file that contains the dynamic block definition, or open the drawing file that has been saved as a block.
- 2 Follow the steps in To open an existing block definition in the Block Editor.

Standard toolbar



Command line: OPEN

Shortcut menu: Right-click a selected block. Click Block Editor.

To delete an element from a dynamic block definition

- 1 In the Block Editor, select the element you want to delete from the dynamic block definition.
- 2 Press the DELETE key.

To modify an action's selection set in a dynamic block definition

- 1 In the Block Editor, select an action.
- 2 On the command line, enter **bactionset**, and then press ENTER.
- 3 Follow the prompts on the command line.
- 4 On the Block Editor toolbar, click Save Block Definition.
- 5 (Optional) If you are finished using the Block Editor, click Close Block Editor.

To redefine a stretch or polar stretch action's stretch frame set in a dynamic block definition

- 1 In the Block Editor, double-click a stretch or polar stretch action.
- 2 Follow the prompts on the command line to define a new stretch frame and add objects to or remove objects from the action's selection set.
- 3 On the Block Editor toolbar, click Save Block Definition.
- 4 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Command line: BACTIONSET

Correct Errors in Dynamic Block Definitions

A yellow alert icon is displayed in the Block Editor if a dynamic block definition contains errors or is incomplete. You need to correct the errors (or complete the block) so the block reference will function properly in a drawing.

The yellow alert icon indicates that the block definition is not defined correctly or is incomplete. For example, a parameter that is not associated with an action will display an alert icon. An action that is not associated with a parameter or a selection set, will also display an alert icon.

You can correct these errors by double-clicking the yellow alert icon and following the prompts on the command line.

After you create a dynamic block definition in the Block Editor, you should save it and then test the functionality of the block reference in a drawing. If the block reference isn't functioning the way you intended, open the definition in the Block Editor and double-check the types of parameters and actions you used as well as their dependencies and properties.

To correct an error indicated by a yellow alert icon in a dynamic block definition

- 1 In the Block Editor, double-click a yellow alert icon.
- 2 Follow the prompts on the command line to correct the errors in the dynamic block definition.
- 3 On the Block Editor toolbar, click Save Block Definition.
- 4 (Optional) If you are finished using the Block Editor, click Close Block Editor.

To view dependencies within a dynamic block definition

- 1 In the Block Editor, select a parameter or an action.
If you selected a parameter, its associated action is highlighted.
If you selected an action, its associated parameter and selection set of geometry is highlighted.
- 2 When you're finished viewing the dependencies, press ESC.
- 3 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Save a Block in the Block Editor

When you are finished adding elements to your dynamic block definition in the Block Editor, save the block definition.

In the Block Editor, you can save your block definition by clicking the Save Block Definition button on the Block Editor toolbar, or by entering **bSAVE** on the command line. You should then save your drawing to make sure that the block definition is saved in the drawing.

When you save a block definition in the Block Editor, the current values of the geometry and parameters in the block are set as the default values for the block reference. When you create a dynamic block that uses visibility states, the default visibility state for the block reference is the visibility state at the top of the list in the Manage Visibility States dialog box.

Once you've saved the block definition, you can close the Block Editor and try your block in a drawing.

NOTE If you click File ► Save while you are in the Block Editor, you will save the drawing but not the block definition. You must specifically save the block definition while you are in the Block Editor.

To save a block definition in the Block Editor

- 1 On the Block Editor toolbar, click Save Block Definition.
- 2 Click File ► Save to save the block definition in the drawing.
- 3 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar



Command line: BSAVE

To save a copy of the current block definition in the Block Editor under a new name

- 1 On the Block Editor toolbar, click Save Block Definition As.
- 2 In the Save Block As dialog box, enter a name for the new block definition.
- 3 Click OK.
- 4 Click File ► Save to save the block definition in the drawing.
- 5 (Optional) If you are finished using the Block Editor, click Close Block Editor.

Standard toolbar



Command line: BSAVEAS

Control the Color and Linetype Properties in Blocks

The objects in an inserted block can retain their original properties, can inherit properties from the layer on which they are inserted, or can inherit the properties set as current in the drawing.

Generally when you insert a block, the color, linetype, and linewidth 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 linewidth settings. These objects have floating properties.

You have three choices for how the color, linetype, and linewidth properties of objects are treated when a block reference is inserted.

- Objects in the block do not inherit color, linetype, and linewidth 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 linewidth properties individually for each object in the block definition: do not use BYBLOCK or BYLAYER color, linetype, and linewidth settings when creating these objects.

- Objects in the block inherit color, linetype, and linewidth properties from the color, linetype, and linewidth 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 linewidth to BYLAYER.

- Objects inherit color, linetype, and linewidth properties from the current color, linetype, and linewidth that you have set explicitly, that is, that you have set to override the color, linetype, or linewidth assigned to the current layer. If you have not explicitly set them, then these properties are inherited from the color, linetype, and linewidth assigned to the current layer.

For this choice, before you create objects to be included in the block definition, set the current color or linetype to BYBLOCK.

If you want objects in a block to	Create objects on these layers	Create objects with these properties
Retain original properties	Any but 0 (zero)	Any but BYBLOCK or BYLAYER
Inherit properties from the current layer	0 (zero)	BYLAYER
Inherit individual properties first, then layer properties	Any	BYBLOCK

Floating properties also apply to nested blocks when the nested block references and the objects they contain use the settings required for floating properties.

To set the color for all new objects

- 1 On the Properties toolbar, click the Color control.
- 2 Either click a color to draw all new objects in that color, or click Select Color to display the Select Color dialog box and do one of the following:
 - On the Index tab, click a color or enter the ACI color number (1-255) or name in the Color box, and then click OK.
 - On the True Color tab, select the HSL color model in the Color Model option and specify a color by entering a color value in the Color box or by specifying values in the Hue, Saturation, and Luminance boxes, and then click OK.
 - On the Color Books tab, select a color book from the Color Book box, select a color by navigating the color book (using the up and down arrows) and clicking on a color chip, and then click OK.
 - Click BYLAYER to draw new objects in the color assigned to the current layer.
 - Click BYBLOCK to draw new objects in the current color until they are grouped into a block. When the block is inserted into the drawing, the objects in the block acquire the current color setting.
- 3 Click OK.

The Color control displays the current color.

Command line: COLOR

To set the linetype for all new objects

- 1 On the Format menu, click Linetype.

- 2** If you need to load additional linetypes, click Load, select one or more linetypes, and click OK.

You can hold down CTRL to select several linetypes or SHIFT to select a range of linetypes.

- 3** In the Linetype Manager, do one of the following:

- Select a linetype and select Current to draw all new objects with that linetype.
- Select BYLAYER to draw new objects in the linetype assigned to the current layer.
- Select BYBLOCK to draw new objects in the current linetype until they are grouped into a block. When the block is inserted in a drawing, the objects in the block acquire the current linetype setting.

- 4** Click OK.

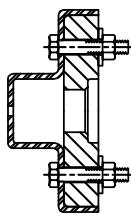
The Linetype control displays the current linetype. If the linetype you want to use is already loaded, you can click the Linetype control and click the linetype to make it current.

Command line: LINETYPE

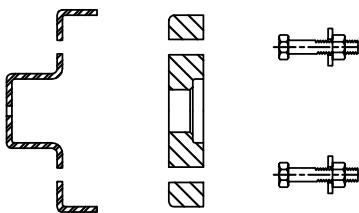
Nest Blocks

Block references that contain other blocks are known as nested blocks. Using blocks within blocks can simplify the organization of a complex block definition.

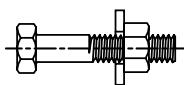
With nested blocks, you can build a single block out of several components. For example, you can insert as a block a drawing of a mechanical assembly that contains a housing, a bracket, and fasteners in which each fastener is a block composed of a bolt, washer, and nut. The only restriction on nested blocks is that you cannot insert blocks that reference themselves.



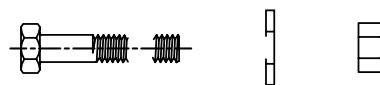
assembly block



blocks that are components of the assembly block



fastener block

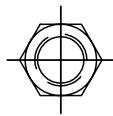


blocks that are components of the fastener block

No procedures for this topic.

Insert Blocks

You can specify the scale of a block reference using different *X*, *Y*, and *Z* values. Inserting a block creates an object called a block reference because it references a block definition stored in your current drawing. An orange lightning bolt icon displayed in the bottom-right corner of the block preview indicates that the block is dynamic.



default values



X scale = .5
Y scale = 1



X scale = 1
Y scale = .5



rotation angle
= 45

If you insert a block that uses different drawing units than the units specified for the drawing, the block is automatically scaled by a factor equivalent to the ratio between the two units.

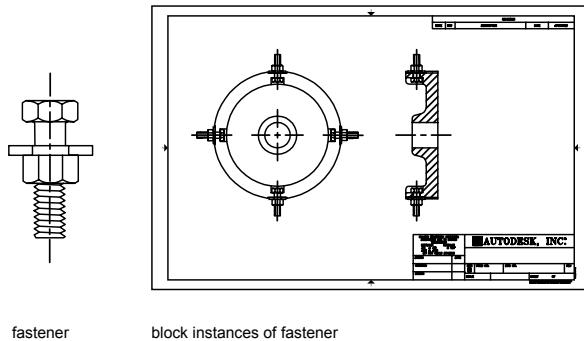
If you insert a block reference that includes editable custom properties or attributes, you can change the values of these custom properties and attributes in the Properties palette while you insert the block. The block's custom

properties and attributes become editable in the Properties palette after the block is specified in one of the following ways:

- In the Insert Dialog Box after clicking OK.
- Using the INSERT Command Line after entering the block name.
- Clicking a block tool on a tool palette.

Insert a Drawing File as a Block

When you insert an entire drawing file into another drawing, the drawing information is copied into the block table of the current drawing as a block definition. Subsequent insertions reference the block definition with different position, scale, and rotation settings, as shown in the following illustration.



fastener

block instances of fastener

Xrefs contained in a drawing you insert may not be displayed properly unless the xref was previously inserted or attached to the destination drawing.

Insert Blocks from Tool Palettes

You can insert blocks from tool palettes by dragging the block tool into the drawing or by clicking the block tool and then specifying an insertion point.

You can choose to be prompted at the command line for a rotation angle (starting from 0) when you click and place the block. When you select this option, the angle that is specified under Rotation in the Tool Properties dialog box is ignored. The prompt for a rotation angle is not shown if you drag the block or xref or, if at the initial insertion command line prompt, you enter **rotate**.

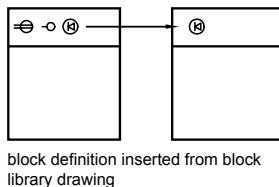
Blocks that are placed by dragging from a tool palette must often be rotated or scaled after they are placed. You can use object snaps when dragging blocks from a tool palette; however, grid snap is suppressed during dragging.

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

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

Insert Blocks from Block Libraries

You can insert one or more block definitions from an existing drawing file into your current drawing file. Choose this method when retrieving blocks from block library drawings. A block library drawing contains block definitions of symbols with similar functions. These block definitions are stored together in a single drawing file for easy accessibility and management.



Insert Blocks with DesignCenter

Use DesignCenter to insert blocks from the current drawing or from another drawing. Drag and drop the block names for quick placement. Double-click the block names to specify the precise location, rotation, and scale of the blocks.

You cannot add blocks to a drawing while another command is active, and you can only insert or attach one block at a time.

Insert Blocks at Intervals

You can insert blocks at intervals along a selected geometric object.

- Use MEASURE to insert a block at measured intervals.

- Use DIVIDE to insert a block at proportional (evenly spaced) intervals.

See also:

- "Create Drawing Files for Use as Blocks" on page 464
- "Overview of Blocks" on page 460
- "Create Block Libraries" on page 463
- Work with Dynamic Blocks in Drawings
- "Add Content with DesignCenter" on page 49
- "Create and Use Tools from Objects and Images" on page 80

To insert a block defined in the current drawing

- 1 On the Insert menu, click Block.
- 2 In the Insert dialog box, in the Name box, select a name from a list of block definitions.
- 3 If you want to use the pointing device to specify the insertion point, scale, and rotation, select Specify On-Screen. Otherwise, enter values in the Insertion Point, Scale, and Rotation boxes.
- 4 If you want the objects in the block to be inserted as individual objects instead of as a single block, select Explode.
- 5 Click OK.

Insert toolbar



Command line: INSERT

To insert a drawing file as a block by dragging

- 1 From Windows Explorer or any folder, drag the drawing file icon into the drawing area.
When you release the button, you are prompted for an insertion point.
- 2 Specify the insertion point and scale and rotation values.

To insert a block using DesignCenter

- 1 On the Tools menu, click DesignCenter.
The DesignCenter window is displayed.

- 2** Do one of the following to list the content you want to insert:
 - On the DesignCenter toolbar, click Tree View Toggle. Click the folder that contains the drawing you want to insert.
 - Click the icon of a drawing file displayed in the tree view.
- 3** Do one of the following to insert the content:
 - Drag the drawing file or block into your current drawing. Use this option when you want to insert blocks quickly and move or rotate the blocks to their precise locations later.
 - Double-click the drawing file or block that you want to insert into your current drawing. Use this option when you want to specify the exact placement, rotation, and scale of the block as you insert it. Use this option also when you want to update a block reference in your drawing from the original source drawing file.

Standard toolbar

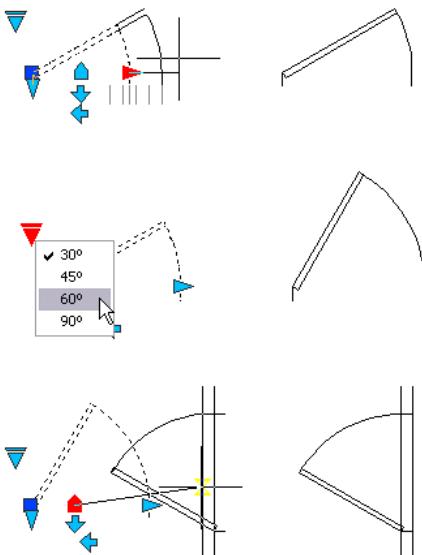


Command line: ADCENTER

Work with Dynamic Blocks in Drawings

A *dynamic* block has flexibility and intelligence. A dynamic block reference can easily be changed in a drawing while you work. You can manipulate the geometry through custom grips or custom properties. This allows you to adjust the block reference in-place as necessary rather than searching for another block to insert or redefining the existing one.

For example, if you insert a door block reference in a drawing, you might need to change the size of the door while you're editing the drawing. If the block is dynamic and defined to have an adjustable size, you can change the size of the door simply by dragging the custom grip or by specifying a different size in Properties palette. You might also need to change the open angle of the door. The door block might also contain an alignment grip, which allows you to align the door block reference easily to other geometry in the drawing.



A dynamic block can have custom grips and custom properties. Depending on how the block was defined, you may be able to manipulate the block through these custom grips and custom properties. By default, custom grips for a dynamic block are a different color than standard grips. You can change the display color for custom grips with the GRIPDYNCOLOR system variable. The following table shows the different types of custom grips that can be included in a dynamic block.

Grip Type	How the Grip Can Be Manipulated in a Drawing
Standard	Within a plane in any direction
Linear	Back and forth in a defined direction or along an axis
Rotation	Around an axis
Flip	Clicked to flip the dynamic block reference
Alignment	Within a plane in any direction; when moved over an object, triggers the block reference to align with the object

Grip Type	How the Grip Can Be Manipulated in a Drawing	
Lookup		Clicked to display a list of items

After you've manipulated a dynamic block in a drawing, you can reset it. When you reset a block reference, the block changes back to the default specified in the block definition. If you non-uniformly scale or explode a dynamic block reference, it loses its dynamic properties. You can reset the block to its default values, which will make it dynamic again.

Some dynamic blocks are defined so that geometry within the block can only be edited to certain sizes specified in the block definition. When you use a grip to edit the block reference, tick marks are displayed at the locations of valid values for the block reference. If you change a block property value to a value other than one specified in the definition, the parameter will adjust to the closest valid value. For example, a block is defined to have a length of 2, 4, and 6. When you try to change that distance value to 10, the resulting value is 6 because it is the closest valid value.

To manipulate a dynamic block using custom grips

- 1 In a drawing, select a dynamic block reference.
- 2 Use the grips to stretch or change the block.

To manipulate a dynamic block using custom properties

- 1 In a drawing, select a dynamic block reference.
- 2 In the Properties palette, under Custom, change the required values.

To reset block references in a drawing

- 1 In a drawing, select a dynamic block reference.
- 2 Right-click in the drawing area. Click Reset Block.

NOTE To reset more than one block reference at a time, use the **RESETBLOCK** command.

Command line: **RESETBLOCK**

To change the display color for custom grips

- 1 On the command line, enter **gripdyncolor**.

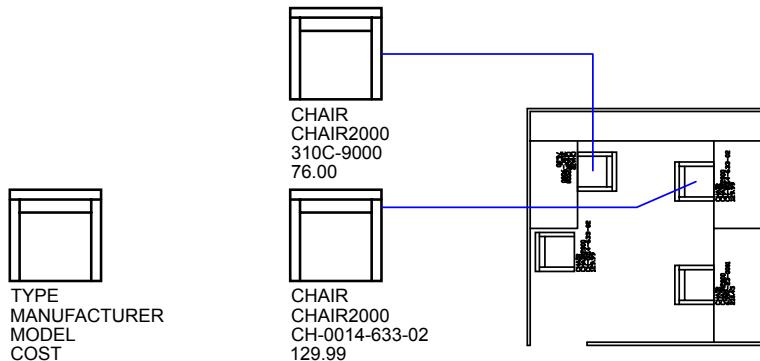
- 2 Enter an integer from 1 to 255 (ACI color).
- 3 Press ENTER.

Attach Data to Blocks (Block Attributes)

You can attach information to blocks and later extract the information to create a bill of materials or other report.

Overview of Block Attributes

An attribute is a label or tag that attaches data to a block. Examples of data that might be contained in an attribute are part numbers, prices, comments, and owners' names. The tag is equivalent to a column name in a database table. The following illustration shows tags for type, manufacturer, model, and cost.



Attribute information extracted from a drawing can be used in a spreadsheet or database to produce a parts list or a bill of materials. You can associate more than one attribute with a block, provided that each attribute has a different tag.

Whenever you insert a block that has a variable attribute, you are prompted to enter data to be stored with the block. Blocks may also use constant attributes, attributes whose values do not change. Constant attributes do not prompt you for a value when you insert the block.

Attributes also can be "invisible." An invisible attribute is not displayed or plotted; however, the attribute information is stored in the drawing file and can be written to an extraction file for use in a database program.

See also:

"[Modify Block Attributes](#)"

Define Block Attributes

To create an attribute, you first create an attribute definition, which describes the characteristics of the attribute. The characteristics include the tag (which is a name that identifies the attribute), the prompt displayed when you insert the block, value information, text formatting, location, and any optional modes (Invisible, Constant, Verify, and Preset).

After creating the attribute definition, you select it as one of the objects when you define the block. Then, whenever you insert the block, you are prompted with the text you specified for the attribute. For each new block insertion, you can specify a different value for the attribute.

To use several attributes together, define them and then include them in the same block. For example, you can define attributes tagged "Type," "Manufacturer," "Model," and "Cost," and then include them in a block called CHAIR.



If you plan to extract the attribute information for use in a parts list, you may want to keep a list of the attribute tags you have created. You will need this tag information later when you create the attribute template file.

Correct Mistakes in Block Attribute Definitions

If you make a mistake, you can use the Properties palette or DDDEDIT to edit an attribute definition before it is associated with a block. You can change the tag, prompt, and default value.

Attach Attributes to Blocks

You can attach attributes to a block when you define or redefine that block. When you are prompted to select the objects to include in the block definition, include in the selection set any attributes you want to attach to the block.

To attach several attributes to the same block, first define the attributes and then include them in the block definition. The order in which you select the attributes determines the order in which you are prompted for attribute information when you insert the block.

Usually, the order of the attribute prompts is the same as the order in which you selected the attributes when you created the block. However, if you used crossing or window selection to select the attributes, the order of the prompts is the reverse of the order in which you created attributes. You can use the Block Attribute Manager to change the order in which you are prompted for attribute information when you insert the block reference.

When you're working in the Block Editor, you can also use the Attribute Order dialog box to change the order in which you are prompted for attribute information when you insert the block reference. You can only do this when you have a block definition open in the Block Editor.

Use Attributes Without Attaching Them to Blocks

Stand-alone attributes can also be created. Once attributes have been defined, and the drawing is saved, this drawing file can be inserted into another drawing. When the drawing is inserted, you are prompted for the attribute values.

To create an attribute definition

- 1** On the Draw menu, click Block ► Define Attributes.
- 2** In the Attribute Definition dialog box, set the attribute modes and enter tag information, location, and text options.
- 3** Click OK.

After creating the attribute definition, you can select it as an object while creating a block definition. If the attribute definition is incorporated into a block, whenever you insert the block, you are prompted with the text string you specified for the attribute. Each subsequent instance of the block can have a different value specified for the attribute.

Command line: ATTDEF

To edit an attribute definition before it is associated with a block

- 1 On the Modify menu, click Object ▶ Text.
- 2 Select the attribute to edit.
- 3 In the Edit Attribute Definition dialog box, specify the attribute tag, prompt, and default value. Then click OK.

Command line: DDEDIT

To change the order of attribute definitions

- 1 In the Block Editor, select a block attribute.
- 2 Right-click in the Block Editor drawing area.
- 3 Click Attribute Order.
- 4 In the Attribute Order dialog box, select an attribute definition.
- 5 Click Move Up or Move Down to change the order of the attribute definitions.
- 6 Repeat steps 2 and 3 until the attribute definition list is in the desired order.
- 7 Click OK.

Command line: BEDIT ▶ BATTORDER

Extract Data from Block Attributes

If you have attached attributes to blocks, you can then query one or more drawings for this block attribute information and save it to a table or to an external file.

Extracting attribute information is an easy way to produce a schedule or bill of materials directly from your drawing data. For example, a facilities drawing might contain blocks representing office equipment. If each block has attributes identifying the model and manufacturer of the equipment, you can generate a report that estimates the cost of the equipment.

The Attribute Extraction wizard guides you through selecting drawings, block instances, and attributes. The wizard can also create a file with a .blk file extension that contains all the settings for later reuse.

Output to a Table

If you extract attribute data to a table, the table is inserted in the current drawing and current space (model space or paper space) and on the current layer.

When you update the table, the attribute information is extracted again and the data rows in the table are replaced. If you have included a title row or one or more header rows in the table, they are not replaced during update.

NOTE For access to shortcut menus in the drawing area that are needed for editing and updating tables, the Shortcut Menus in Drawing Area must be checked in the Options dialog box, User Preferences tab.

Output to a File

If you save the data to an external file, the comma-separated (CSV), tab-separated (TXT), Microsoft Excel (XLS), and Microsoft Access (MDB) file formats are available.

When the characters period (.), comma (,), or pound sign (#) are written to an Excel or Access file, they are replaced with their Unicode representation.

To extract block attributes to a table or a file

- Click Tools menu > Attribute Extraction.

The Attribute Extraction Wizard is displayed. The wizard provides step-by-step instructions for extracting information from block attributes in the current drawing or other drawings. The information is used to create a table in the current drawing or is saved to an external file.

The “Attribute Extraction Wizard” topic describes the options in the wizard.

Modify II toolbar



Command line: EATTEXT

To update an attribute extraction table

- When the “Attribute Extraction Table Needs Updating” notification is displayed in the status tray, click Refresh Table Data.

To turn off update notification for attribute extraction tables

- 1 To turn off notification for all notification-enabled tables in the drawing, right-click the notification icon in the status tray. Clear the Enable Notifications for Out-of-Synch Table Data option.
- 2 To turn off notification for a selected table, right-click. Click the Suppress Notifications for This Table option.

The notification option is on by default on the Table Style page when you use the Attribute Extraction wizard.

Extract Block Attribute Data (Advanced)

You can extract attribute information from a drawing and create a separate text file for use with database software. This feature is useful for creating parts lists with information already entered in the drawing database. Extracting attribute information does not affect the drawing.

To create a parts list

- Create and edit an attribute definition
- Enter values for the attributes as you insert the blocks
- Create a template file and then extract attribute information to a text file

To extract attribute information, you first create an attribute template file using any text processor, then generate the attribute extraction file using AutoCAD, and, finally, open the attribute extraction file in a database application. If you plan to extract the attribute information to a DXF (drawing interchange format) file, it is not necessary to first create an attribute template file.

NOTE Make sure that the attribute extraction file does not have the same name as the attribute template file.

Create an Attribute Extraction Template File

Before you extract attribute information, you must create an ASCII template file to define the structure of the file that will contain the extracted attribute information. The template file contains information about the tag name, data type, field length, and number of decimal places associated with the information you want to extract.

Each field in the template file extracts information from the block references in the drawing. Each line in the template file specifies one field to be written

to the attribute extraction file, including the name of the field, its character width, and its numerical precision. Each record in the attribute extraction file includes all the specified fields in the order given by the template file.

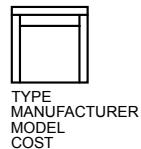
The following template file includes the 15 possible fields. *N* means numeric, *C* means character, *www* means a 3 digit number for the total width of the field, and *ddd* means a 3 digit number representing how many numeric decimal places are to be displayed to the right of the decimal point.

BL:NAME 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*)
numeric Nwwwddd (Numeric attribute tag)
character Cwww000 (Character attribute tag)

The template file can include any or all of the BL:xxxxxx field names listed, but must include at least one attribute tag field. The attribute tag fields determine which attributes, hence which blocks, are included in the attribute extraction file. If a block contains some, but not all, of the specified attributes, the values for the absent ones are filled with blanks or zeros, depending on whether the field is a character field or a numeric field.

Comments should not be included in an attribute template file.

The illustration and table show an example of the type of information you're likely to extract, including block name, manufacturer, model number, and cost.



Field	(C)haracter or (N)umeric data	Maximum field length	Decimal places
Block name	C	040	000
Manufacturer	C	006	000
Model	C	015	000
Cost	N	006	002

You can create any number of template files, depending on how you'll use the data. Each line of a template file specifies one field to be written in the attribute extraction file.

Follow these additional guidelines:

- Be sure to place a space between the attribute tag and the character or numeric data. Use SPACEBAR, not TAB, to enter the space.
- Press ENTER at the end of each line, including the last line.
- Each attribute extraction template file must include at least one attribute tag field, but the same field can appear only once in the file.

The following is a sample template file.

BL:NAME C008000 (*Block name, 8 characters*)
 BL:X N007001 (*X coordinate, format nnnnnn.d*)
 BL:Y N007001 (*Y coordinate, format nnnnnn.d*)
 SUPPLIER C016000 (*Manufacturer's name, 16 characters*)
 MODEL C009000 (*Model number, 9 characters*)
 PRICE N009002 (*Unit price, format nnnnnnnn.dd*)

NOTE The format code for a numeric field includes the decimal point in the total field width. For example, the minimum field width to accommodate the number 249.95 would be 6 and would be represented as N006002. Character fields do not use the last three digits of the format code.

Create an Attribute Extraction File

After creating a template file, you can extract the attribute information using one of the following formats:

- Comma-delimited format (CDF)

- Space-delimited format (SDF)
- Drawing interchange format (DXF)

The CDF format produces a file containing one record for each block reference in a drawing. A comma separates the fields of each record, and single quotation marks enclose the character fields. Some database applications can read this format directly.

The SDF format also produces a file containing one record for each block reference in a drawing. The fields of each record have a fixed width and employ neither field separators nor character-string delimiters. The dBASE III Copy . . . SDF operation also produces SDI-format files. The Append From... SDF operation can read a file in dBASE IV format, which user programs written in FORTRAN can easily process.

DXF produces a subset of the drawing interchange format containing only block reference, attribute, and end-of-sequence objects. This option requires no attribute extraction template. The file extension .dxx distinguishes an extraction file in DXF format from normal DXF files.

Use the Attribute Extraction File

The attribute extraction file lists values and other information for the attribute tags you specified in the template file.

If you specified a CDF format using the sample template, the output might appear as follows:

```
'DESK', 120.0, 49.5, 'ACME INDUST.', '51-793W', 379.95
'CHAIR', 122.0, 47.0, 'ACME INDUST.', '34-902A', 199.95
'DESK', -77.2, 40.0, 'TOP DRAWER INC.', 'X-52-44', 249.95
```

By default, character fields are enclosed with single quotes (apostrophes). The default field delimiter is a comma. The following two template records can be used to override these defaults:

```
C:QUOTE c (Character string delimiter)
C:DELIM c (Field delimiter)
```

The first nonblank character following the C:QUOTE or C:DELIM field name becomes the respective delimiter character. For example, if you want to enclose character strings with double quotes, include the following line in your attribute extraction template file:

```
C:QUOTE "
```

The quote delimiter must not be set to a character that can appear in a character field. Similarly, the field delimiter must not be set to a character that can appear in a numeric field.

If you specified an SDF format using the sample template, the file might be similar to the following example.

(NAME)	(X)	(Y)	(SUPPLIER)	(MODEL)	(PRICE)
DESK	120.0	49.5	ACME INDUST.	51-793W	379.95
CHAIR	122.0	47.0	ACME INDUST.	34-902A	199.95
DESK	-77.2	40.0	TOP DRAWER INC.	X-52-44	249.95

The order of the fields corresponds to the order of the fields in the template files. You can use these files in other applications, such as spreadsheets, and you can sort and manipulate the data as needed. For example, you can open an attribute extraction file in Microsoft Excel in which you can specify a separate column for each field. See the documentation for your spreadsheet program for information about how to use data from other applications. If you open the file in Notepad or another Windows text processor, you can paste the information back into the drawing as text.

Nested Blocks

The line BL:LEVEL in a template file reports the nesting level of a block reference. A block that is inserted in a drawing has a nesting level of 1. A block reference that is part of (nested within) another block has a nesting level of 2, and so on.

For a nested block reference, the X,Y, Z coordinate values, scale factors, extrusion direction, and rotation angle reflect the actual location, size, orientation, and rotation of the nested block in the world coordinate system.

In some complex cases, nested block references cannot be correctly represented with only two scale factors and a rotation angle, for example, if a nested block reference is rotated in 3D. When this happens, the scale factors and rotation angle in the extracted file record are set to zero.

Error Handling

If a field is not wide enough for the data that is to be placed in it, the data is truncated and the following message is displayed:

**** Field overflow in record <record number>**

This could happen, for example, if you have a BL:NAME field with a width of 8 characters and a block in your drawing has a name 10 characters long.

To create an attribute extraction template file

- 1 Start Notepad.

You can use any text editor or word processor that can save a text file in ASCII format.

- 2 Enter template information in Notepad. See “Extract Block Attribute Data (Advanced)” on page 568 for format information.
- 3 Save the file with a .txt file extension.

To extract data about a specific tag, insert the tag name in place of the "numeric" or "character" fields.

WARNING Do not use tab characters when constructing the template file with a word processor. If you use tab character alignment, the attribute information file is not created. To align the columns, insert ordinary spaces by pressing SPACEBAR. The use of tab characters may cause inconsistent alignment.

To extract attribute information

- 1 At the Command prompt, enter **attext**.
- 2 In the Attribute Extraction dialog box, specify the appropriate file format: CDF, SDF, or DXF.
- 3 Specify the objects to extract attributes from by choosing Select Objects. You can select a single block or multiple blocks in the drawing.
- 4 Specify the attribute template file to use by entering the file name or by choosing Template File and browsing.
- 5 Specify the output attribute information file by entering the file name or by choosing Output File and browsing.
- 6 Click OK.

Command line: ATTEXT

Modify Blocks

You can modify a block definition or a block reference already inserted in the drawing.

Modify a Block Definition

You can redefine block definitions in your current drawing. Redefining a block definition affects both previous and future insertions of the block in the current drawing and any associated attributes.

There are two methods for redefining a block definition:

- Modify the block definition in the current drawing.

- Modify the block definition in the source drawing and reinsert it into the current drawing.

The method you choose depends on whether you want to make changes in the current drawing only or in a source drawing also.

Modify a Block Definition in the Current Drawing

To modify a block definition, follow the procedure to create a new block definition, but enter the name of the existing block definition. This replaces the existing block definition, and all the references to that block in the drawing are immediately updated to reflect the new definition.

To save time, you can insert and explode an instance of the original block and then use the resulting objects in creating the new block definition.

Update a Block Definition That Originated from a Drawing File

Block definitions created in your current drawing by inserting a drawing file are not updated automatically when the original drawing is modified. You can use INSERT to update a block definition from the drawing file.

Update a Block Definition That Originated from a Library Drawing (Advanced)

DesignCenter™ does not overwrite an existing block definition in a drawing with one that comes from another drawing. To update a block definition that came from a library drawing, use WBLOCK to create a separate drawing file from the library drawing block. Then, use INSERT to overwrite the block definition in the drawing that uses the block.

NOTE Block descriptions are stripped off when using INSERT. Use the Clipboard to copy and paste a block description displayed in the Block Definition dialog box from one block definition to another.

Modify the Description of a Block

To modify the DesignCenter description of a block definition, use BLOCK. You can also add descriptions to any number of existing blocks in the Block Definition dialog box.

Redefine Block Attributes

You can attach attributes to a block when you define or redefine that block. When you are prompted to select the objects to include in the block definition, include the desired attributes in the selection set. Redefining the attributes in

the block definition has the following effects on block references that were previously inserted:

- Constant attributes, which have a fixed value, are lost and replaced by any new constant attributes.
- Variable attributes remain unchanged, even if the new block definition has no attributes.
- New attributes do not appear in the existing block references.

See also:

["Attach Data to Blocks \(Block Attributes\)" on page 563](#)

["Modify Dynamic Block Definitions" on page 549](#)

To update a block definition that originated from a drawing file

- 1 On the Tools menu, click DesignCenter.
- 2 In the tree view, click the folder that contains the drawing file from which the block originated.
- 3 In the content area (on the right side), right-click the drawing file.
- 4 On the shortcut menu, click Insert as Block.
- 5 In the Insert dialog box, click OK.
- 6 In the alert box, click Yes to overwrite the existing block definition.
- 7 Press ESC to exit the command.

To modify a block description

- 1 On the Modify menu, click Object ▶ Block Description.
- 2 In the Block Definition dialog box, in the Name list, select the block for which you want to modify the block description.
- 3 In the Description box, enter or modify the description of the block.
- 4 Click OK.
- 5 A message box displays the warning "Block name is already defined. Do you want to re-define it?" Click Yes to redefine the block.

Command line: BLOCK

Modify Block Attributes

You can modify attributes in block definitions with the Block Attribute Manager. For example, you can modify the following:

- Properties that define how values are assigned to an attribute and whether or not the assigned value is visible in the drawing area
- Properties that define how attribute text is displayed in the drawing
- Properties that define the layer that the attribute is on and the attribute line's color, weight, and type

By default, attribute changes you make are applied to all existing block references in the current drawing.

Changing the attribute properties of existing block references does not affect the values assigned to those blocks. For example, in a block containing an attribute whose tag is Cost and value is 19.99, the 19.99 value is unaffected if you change the tag from Cost to Unit Cost.

Updating attributes with duplicate tag names can lead to unpredictable results. Use the Block Attribute Manager to find duplicate tags and change tag names.

If constant attributes or nested attributed blocks are affected by your changes, use REGEN to update the display of those blocks in the drawing area.

Change the Prompt Order for Attribute Values

When you define a block, the order in which you select the attributes determines the order in which you are prompted for attribute information when you insert the block. You can use the Block Attribute Manager to change the order of prompts that request attribute values.

Remove Block Attributes

You can remove attributes from block definitions and from all existing block references in the current drawing. Attributes removed from existing block references do not disappear in the drawing area until you regenerate the drawing using REGEN.

You cannot remove all attributes from a block; at least one attribute must remain. If you need to remove all attributes, redefine the block.

Update Block References

You can update attributes in all block references in the current drawing with changes you made to the block definition. For example, you may have used the Block Attribute Manager to modify attribute properties in several block

definitions in your drawing but elected not to automatically update existing block references when you made the changes. Now that you are satisfied with the attribute changes you made, you can apply those changes to all blocks in the current drawing.

You can also use ATTSYNC to update attribute properties in block references to match their block definition, or to update a block instance after you redefine a block attribute using BLOCK, -BLOCK, or BEDIT.

Updating attribute properties in block references does not affect any values that have been assigned to those attributes.

Edit Attributes in a Block Reference

You can select an attribute in a block reference and use the Properties palette to change its properties, or you can use the Enhanced Attribute Editor to modify all the attributes in a selected block reference.

See also:

["Define Block Attributes" on page 564](#)

["Modify a Block Definition" on page 573](#)

To edit attributes assigned to a block definition

- 1 On the Modify menu, click Object ▶ Attribute ▶ Block Attribute Manager.
- 2 In the Block Attribute Manager, select a block from the Block list, or click Select Block and select a block in the drawing area.
- 3 In the list of attributes, double-click the attribute you want to edit, or select the attribute and click Edit.
- 4 In the Edit Attribute dialog box, make the attribute changes you want, and then click OK.

Modify II toolbar



Command line: BATTMAN

To specify whether changes are applied to existing block references

- 1 On the Modify menu, click Object ▶ Attribute ▶ Block Attribute Manager.
- 2 In the Block Attribute Manager, click Settings.

- 3** In the Settings dialog box, do one of the following:
 - To apply changes to existing block references, select the Apply Changes to Existing References option.
 - To apply changes only to new block insertions, clear the Apply Changes to Existing References option.
- 4** Click OK.

Modify II toolbar



Command line: BATTMAN

To highlight duplicate attribute tags in a block

- 1** On the Modify menu, click Object ▶ Attribute ▶ Block Attribute Manager.
- 2** In the Block Attribute Manager, click Settings.
- 3** In the Settings dialog box, select Emphasize Duplicate Tags.
- 4** Click OK.

Modify II toolbar



Command line: BATTMAN

To change the prompt order for attribute values

- 1** On the Modify menu, click Object ▶ Attribute ▶ Block Attribute Manager.
- 2** In the Block Attribute Manager, select a block from the Block list, or click Select Block and select a block in the drawing area.
For the selected block, attributes are listed in their prompt order.
- 3** To move an attribute up in the prompt order, select the attribute, and then click Move Up; to move an attribute down in the prompt order, select the attribute, and then click Move Down.

NOTE The Move Up and Move Down buttons are unavailable for attributes with constant values (Mode=C).

Modify II toolbar



Command line: BATTMAN

To remove an attribute from a block definition and all block references

- 1 On the Modify menu, click Object ► Attribute ► Block Attribute Manager.
- 2 In the Block Attribute Manager, select a block from the Block list, or click Select Block and select a block in the drawing area.
- 3 (Optional) If you do not want attributes removed from existing instances of the block, click Settings, and, in the Settings dialog box, clear Apply Changes to Existing References.
- 4 In the Block Attribute Manager, select an attribute from the attribute list, and then click Remove.

Attributes removed from existing block instances do not disappear until you regenerate the drawing using REGEN.

Modify II toolbar



Command line: BATTMAN

To update existing block references with attributes you have modified

- 1 On the Modify menu, click Object ► Attribute ► Block Attribute Manager.
- 2 In the Block Attribute Manager, select a block from the Block list, or click Select Block and select a block in the drawing area.
- 3 Click Sync to update attributes you have changed in all block references for the selected block.

Modify II toolbar



Command line: BATTMAN

To update the attributes in block references for a selected block definition

- 1 On the Modify II toolbar, click Synchronize Attributes.
- 2 At the prompt, do one of the following:
 - Enter **name**, and then enter the name of the block whose block references you want to update.
 - Enter **?** to view a list of blocks, and then enter **name**, followed by the name of the block.
 - Press **ENTER**, and then use your pointing device to select a block in the drawing area.

If you specify a block that does not exist, or if the block exists but does not contain attributes, an error message is displayed.

Modify II toolbar



Command line: ATTSYNC

Alternate

- 1 On the Modify menu, click Object ▶ Attribute ▶ Single.
- 2 In the drawing area, select the block you want to edit.
- 3 In the Enhanced Attribute Editor, select the attribute you want to edit. You can change the attribute value or choose another tab and edit other attribute properties.
- 4 Make the attribute changes you want, and then do one of the following:
 - Click **Apply** to save your changes. The Enhanced Attribute Editor remains open.
If you click **Cancel** later to exit the Enhanced Attribute Editor, attribute changes you made prior to choosing **Apply** are not reversed.
 - Click **OK** to save your changes and close the Enhanced Attribute Editor.

- Click Select Block to edit the attributes of a different block. If you made changes to the current block, but have not saved them, you are prompted to do so before selecting a new block.

Modify II toolbar



Command line: EATTEDIT

Change the Color and Linetype in a Block

Blocks containing objects with floating properties inherit their color and linetype from the layer on which they are inserted. Depending on how the objects in the block were created, blocks can also inherit floating color and linetype properties from the current explicit color and linetype that you set to override the layer settings.

If a block was not created using objects with floating color and linetype properties, the only way to change these properties is to redefine the block.

See also:

[“Control the Color and Linetype Properties in Blocks” on page 553](#)

To change the layer of an object

- 1 Select the objects whose layer you want to change.
- 2 On the Layers toolbar, click the Layer control.
- 3 Select the layer that you want to assign to the objects.

Command line: LAYER

To change the color assigned to a layer

- 1 On the Layers toolbar, click the Layer Properties Manager button.
- 2 In the Layer Properties Manager, click the color you want to change.
- 3 In the Select Color dialog box, use one of the following methods:
 - On the Index tab, click a color and then click OK.
 - On the Index tab, enter the ACI color number (1-255) or name in the Color box, and then click OK.

- On the True Color tab, select the HSL color model in the Color Model option and specify a color by entering a color value in the Color box or by specifying values in the Hue, Saturation, and Luminance boxes, and then click OK.
- On the Color Books tab, select a color book from the Color Book box, select a color by navigating the color book (using the up and down arrows) and clicking on a color chip, and then click OK.

4 Click OK.

Layers toolbar



Command line: LAYER

To change the color of an object, overriding the layer's color

- 1 On the Standard toolbar, click Properties.
- 2 Select the objects whose color you want to change.
- 3 In the Properties palette, select Color.
An arrow is displayed in the right column.
- 4 Click the arrow and select a color from the list.

Standard toolbar



Command line: PROPERTIES

To change the linetype assigned to a layer

- 1 On the Layers toolbar, click the Layer Properties Manager button.
- 2 In the Layer Properties Manager, click Load and click one or more linetypes to load, and then click OK.
You can hold down CTRL to select several linetypes or SHIFT to select a range of linetypes.
- 3 Select a layer from the list and then click Details to expand the dialog box.
- 4 Select a linetype from the Linetype list.

- 5 Click OK to close the dialog box.

You can click Layer on the Format menu to open the Layer Properties Manager.

Layers toolbar



Command line: LAYER

To change the linetype of an object, overriding the layer's linetype

- 1 Select the objects whose linetype you want to change.
- 2 On the Properties toolbar, click the Linetype control.
- 3 Click the linetype that you want to assign to the objects.

Command line: LINETYPE

To modify an existing block definition

- 1 Select the block to modify.
- 2 Right-click the block and click Properties on the shortcut menu.
- 3 In the Properties palette, select and modify X and Y position, scale, rotation values, or other properties.

Command line: PROPERTIES

Disassemble a Block Reference (Explode)

If you need to modify one or more objects within a block separately, you can disassemble, or explode, the block reference into its component objects. After making the changes, you can

- Create a new block definition
- Redefine the existing block definition
- Leave the component objects uncombined for other uses

You can automatically explode block references as you insert them by selecting the Explode option in the Insert dialog box.

To explode a block reference

- 1 On the Modify menu, click Explode.
- 2 Select the block to explode and press ENTER.

The block reference is disassembled into its component objects; however, the block definition still exists in the drawing for insertion later.

Modify toolbar



Command line: EXPLODE

To control properties while you explode an object

- 1 Enter **xplode**.
- 2 Select the objects to be exploded.
- 3 If you selected more than one object, enter **i** to control properties for individual objects, or enter **g** to control properties for all the selected objects.
- 4 Enter an option for a property you want to change.
The property is applied to the compound object and the prompt is redisplayed.
- 5 Enter another option, or enter **e** to explode the selected objects.
The selected objects are exploded and the properties you specified are applied to the component objects.

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 460](#)

To remove a block definition

- 1 On the File menu, click Drawing Utilities ► Purge.

The Purge dialog box displays a tree view of named objects that can be purged.

- 2 To purge blocks, use one of the following methods:

- To purge all unreferenced blocks, select Blocks. To include nested blocks, select Purge Nested Items.
- To purge specific blocks, double-click Blocks to expand the Block tree view. Select the blocks to be purged.

If the item you want to purge is not listed, select View Items You Cannot Purge.

- 3 You are prompted to confirm each item in the list. If you do not want to confirm each purge, clear the Confirm Each Item to Be Purged option.

- 4 Click Purge.

To confirm the purging of each item, respond to the prompt by choosing Yes or No, or Yes to All if more than one item is selected.

- 5 Select more items to purge, or click Close.

Command line: PURGE

Change Existing Objects

2 |

You can select objects, view and edit object properties, and perform general and object-specific editing operations.

In this chapter

- Select Objects
- Correct Mistakes
- Erase Objects
- Use Windows Cut, Copy, and Paste
- Modify Objects
- Modify Complex Objects

Select Objects

You have a wide range of options when you need to select objects for editing operations.

Select Objects Individually

At the Select Objects prompt, you can select one or more objects individually.

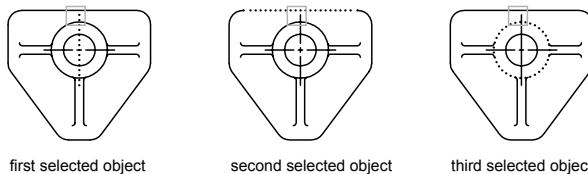
Use the Pickbox Cursor

When the square pickbox cursor is in position to select an object, the object is highlighted. Click to select the object.

You can control the size of the pickbox in the Options dialog box, Selection tab.

Select Objects Close Together

It is difficult to select objects that are close together or lie directly on top of one another. The example shows two lines and a circle that lie within the pickbox.



If selection preview is turned on, you can cycle through the objects by rolling over the object on top to highlight it, and pressing and holding SHIFT and then pressing SPACEBAR continuously. When the required object is highlighted, left-click to select it.

If selection preview is turned off, hold down SHIFT + SPACEBAR and click to cycle through these objects, one after the other, until the one you want is selected. Press ESC to turn off cycling.

Remove Selection from Objects

Remove objects from the current selection set by holding down SHIFT and selecting them again.

See also:

[Select and Modify 3D Subobjects](#)
[Manipulate Composite Solids](#)

To select a single object

- 1 At the Select Objects prompt of any command, move the rectangular pickbox cursor so that the object that you want to select is highlighted.
- 2 Click the object.
The object you selected is highlighted.
- 3 Press ENTER to end object selection.

NOTE If the PICKFIRST system variable is set to 1 (noun-verb selection), you can select objects before entering a command.

To change the size of the pickbox cursor

- 1 Click Tools menu ► Options.
- 2 On the Selection tab, under Pickbox Size, move the pickbox size slider until the pickbox is the size you want to use.
- 3 Click OK.

Command line: DSETTINGS

To cycle through objects for selection

- 1 At the Select Objects prompt, hold down SHIFT + SPACEBAR. Click as near as possible to the object you want.
- 2 Keep clicking until the object you want is highlighted.
- 3 Press ENTER to select the object.

NOTE If selection preview is turned on, you can cycle through the objects by rolling over the object on top to highlight it, and pressing and holding SHIFT and then pressing SPACEBAR continuously. When the required object is highlighted, left-click to select it.

To remove selection from objects

- Hold down SHIFT. Click the objects that you want removed from the selection set.

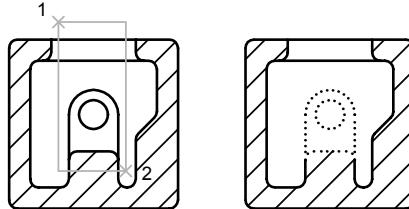
Select Multiple Objects

At the Select Objects prompt, you can select many objects at the same time.

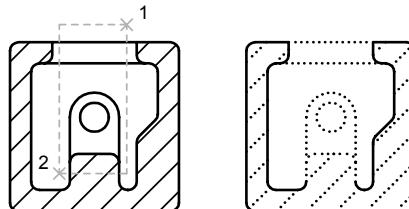
Specify a Rectangular Selection Area

Specify opposite corners to define a rectangular area. The background inside the area changes color and becomes transparent. The direction that you drag your cursor from the first point to the opposite corner determines which objects are selected.

- **Window selection.** Drag your cursor from left to right to select only objects that are entirely enclosed by the rectangular area.
- **Crossing selection.** Drag your cursor from right to left to select objects that the rectangular window encloses or crosses.



objects selected using window selection box

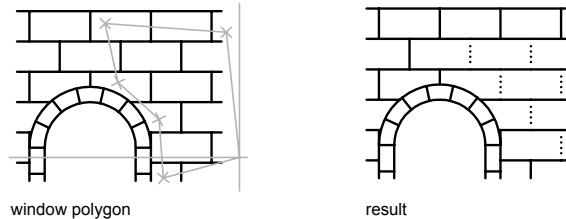


objects selected using crossing selection box

With a window selection, usually the entire object must be contained in the rectangular selection area. However, if an object with a noncontinuous (dashed) linetype is only partially visible in the viewport and all the visible vectors of the linetype can be enclosed within the selection window, the entire object is selected.

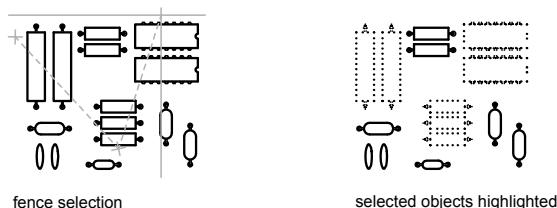
Specify an Irregularly Shaped Selection Area

Specify points to define an irregularly shaped area. Use window polygon selection to select objects entirely enclosed by the selection area. Use crossing polygon selection to select objects enclosed *or* crossed by the selection area.



Specify a Selection Fence

In a complex drawing, use a selection fence. A selection fence looks like a polyline and selects only the objects it passes through. The circuit board illustration shows a fence selecting several components.



Use Other Selection Options

You can see all selection options by entering **?** at the Select Objects prompt. For a description of each of the selection options, see **SELECT**.

Remove Selection from Multiple Objects

You can enter **r** (Remove) at the Select Objects prompt and use any selection option to remove objects from the selection set. If you are using the Remove option and want to return to adding objects to the selection set, enter **a** (Add).

You can also remove objects from the current selection set by holding down SHIFT and selecting them again, or by holding down SHIFT and then clicking and dragging window or crossing selections. You can add and remove objects repeatedly from the selection set.

To see a list of options at the Select Objects prompt

- Enter **?** at the Select Objects prompt.

To select objects within an irregularly shaped area

- 1 At the Select Objects prompt, enter **wp** (Window Polygon).
- 2 Specify points that define an area entirely enclosing the objects you want to select.
- 3 Press ENTER to close the polygon selection area and complete the selection.

To select objects crossing an irregularly shaped area

- 1 At the Select Objects prompt, enter **cp** (Crossing Polygon).
- 2 Specify points that define an area that encloses or crosses the objects you want to select.
- 3 Press ENTER to close the polygon selection area and complete the selection.

To select objects with a fence

- 1 At the Select Objects prompt, enter **f** (Fence).
- 2 Specify points to create a fence that passes through the objects you want to select.
- 3 Press ENTER to complete the selection.

To remove several objects from the selection set

- 1 After selecting objects, at the Select Objects prompt, enter **r** (Remove).
- 2 Enter any selection option such as **cp** (Crossing Polygon) or **f** (Fence), and select the objects to be removed from the selection set.

To return to adding objects to the selection set, enter **a** (Add).

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 lock or unlock a layer

- 1 On the Layers toolbar, click the Layer Properties Manager button.
- 2 In the Layer Properties Manager, click the padlock for the layers that you want to lock.
- 3 Click OK.

If the padlock is closed, the layer is locked and objects on that layer cannot be selected.

Layers toolbar



Command line: LAYER

To lock or unlock a layer by selecting an object on that layer

- 1 Click the Model tab.
- 2 Do either of the following:
 - Click Format menu ► Layer Tools ► Layer Lock
 - Click Format menu ► Layer Tools ► Layer Unlock
- 3 Select an object on the layer you want to lock or unlock.

Layer II toolbar



Command line: LAYLCK, LAYULK

Filter Selection Sets

You can use object properties or object types to include objects in a selection set, or to exclude them.

Using either Quick Select (QSELECT) from the Properties palette or the Object Selection Filters dialog box (FILTER), you can filter selection sets by property (such as color) and by object type. For example, you can select all of the red circles in a drawing without selecting any other object, or you can select all objects except the red circles.

With Quick Select, you can quickly define a selection set based on filtering criteria that you specify, and if an Autodesk or a third-party application was used to add a feature classification to an object, you can select objects by classification property. With object selection filters, you can name and save filters for future use.

With either Quick Select or object selection filters, if you want to filter your selection set based on color, linetype, or linewidth, first consider whether these properties are set to BYLAYER for any objects in your drawing. For example, an object may appear red because its color is set to BYLAYER and the layer color is red.

See also:

[“Customize Object Selection” on page 596](#)

[“Work with Layers” on page 277](#)

To create a selection set using Quick Select

In the following example, you use Quick Select to select the red objects in a drawing.

- 1 Click Tools menu ► Quick Select.
- 2 In the Quick Select dialog box, under Apply To, select Entire Drawing.
- 3 Under Object Type, select Multiple.
- 4 Under Properties, select Color.
- 5 Under Operator, select Equals.
- 6 Under Value, select Red.
- 7 Under How to Apply, select Include in New Selection Set.
- 8 Click OK.

All red objects in the drawing are selected and the Quick Select dialog box closes. Objects that are set to BYLAYER and are red because the layer color is red are not included in the selection set.

NOTE If an application such as Autodesk Map was used to add a feature classification to an object, and the associated classification (XML) file is present,

you can select objects by classification property. Specifically, you can select a classification in the Object Type box and a property in the Properties box.

Command line: QSELECT

To exclude objects from the selection set

You can exclude objects from the current selection set by using the Exclude from New Selection Set option. In the following example, you exclude all circles with a radius greater than 1 from a set of objects already selected.

- 1 With several objects selected, click Tools menu ► Quick Select.
- 2 In the Quick Select dialog box, under Apply To, select Current Selection.
- 3 Under Object Type, select Circle.
- 4 Under Properties, select Radius.
- 5 Under Operator, select Greater Than.
- 6 Under Value, enter **1**.
- 7 Under How to Apply, select Exclude from New Selection Set.
- 8 Click OK.

All circles with a radius greater than 1 are removed from the selection set.

Command line: QSELECT

To append objects to the selection set

You can use Quick Select to append objects to a current selection set. In the following example, you keep the current selection set and append all objects in the drawing that contain hyperlinks whose names begin with **bld1_**.

- 1 Click Tools menu ► Quick Select.
- 2 In the Quick Select dialog box, select Append to Current Selection Set.
- 3 In the Object Type box, select Multiple.
- 4 Under Properties, select Hyperlink.
- 5 Under Operator, select Wildcard Match.
- 6 Under Value, enter **bld1_***.
- 7 Under How to Apply, select Include in New Selection Set.
- 8 Click OK.

Command line: QSELECT

To name and save a filtered list

- 1 At the Command prompt, enter **filter**.
- 2 In the Object Selection Filters dialog box, under Select Filter, select a filter such as **Line**.
- 3 Click Add to List.
- 4 Under Save As, enter a filter name such as **Linefilter**.
- 5 Click Save As.
- 6 Click Apply.

The filter is applied so you can select, in this case, only lines in the drawing. If you select objects with a selection, the filter is applied to all objects in the selection area.

Command line: FILTER

To use a named filter

- 1 At the Select Object prompt, enter '**filter**'. (The apostrophe makes it a transparent command.)
- 2 In the Object Selection Filters dialog box, under Select Filter, select the filter you want to use. Click Apply.
- 3 Use a crossing window to specify objects for selection.

Only the objects selected by the crossing window that match the filter criteria are selected.

Customize Object Selection

You can control several aspects of selecting objects, such as whether you enter a command first or select objects first, the size of the pickbox cursor, and how selected objects are displayed.

For commands that use the Select Objects prompt, you can

- Enter a command first, and then select objects
- Select the objects first, and then enter a command

You can also choose

- Whether objects to be selected are previewed during selection
- Whether selected objects are highlighted
- How you define selection areas and how you create selection sets

Select the Command First

When you use an editing command, a Select Objects prompt is displayed and the crosshairs is replaced with a pickbox. You can respond to the Select Objects prompt in various ways:

- Select objects one at a time.
- Click an empty area. Drag the cursor to define a rectangular selection area.
- Enter a selection option. Enter **?** to display all selection options.
- Combine selection methods. For example, to select most of the objects in the drawing area, select all objects and then remove the objects that you do not want selected.
- Enter '**filter**' to use a named selection filter. The apostrophe runs the command transparently.

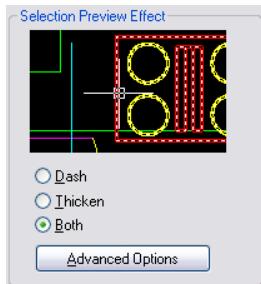
Select Objects First

You can use one of two methods to select objects before starting a command:

- Use the **SELECT** command, and enter **?** to display all selection options. All objects selected are put into the Previous selection set. To use the Previous selection set, enter **p** at the Select Objects prompt of any subsequent command.
- When noun/verb selection is turned on, select objects at the Command prompt before entering a command such as **MOVE**, **COPY**, or **ERASE**. With this method, you can only select objects by clicking them individually or by using automatic selection.
- Enter **qselect** to filter the selection. Then enter **p** at the Select Objects prompt of any subsequent command.

Highlight Objects to Be Selected

Objects are highlighted when the pickbox cursor rolls over them, providing a preview of which object will be selected when you click.



When you specify an area to select multiple objects, the background of the area becomes transparent.

These selection previewing effects are turned on by default. You can turn them off or change the appearance of selection previewing (Options dialog box, Selection tab). When the PICKBOX system variable is set to 0, selection previewing of objects is not available.

Control the Appearance of Selected Objects

By default, selected objects are displayed with dashed lines. You can increase program speed by setting the HIGHLIGHT system variable to 0. Turning off selection highlighting does *not* affect grips on selected objects.

Set Up Default Selection Methods

Options on the Selection tab of the Options dialog box control default selection methods:

- Use selection previewing and selection area effects to preview selection.
- Select objects before entering a command (noun-verb selection) or after entering a command. (PICKFIRST)
- Press SHIFT to append objects to the selection set. (PICKADD)
- Click and drag to create a selection window. Otherwise you must click twice to define the corners of a selection window. (PICKDRAG)
- Start Window or Crossing selection automatically when you click an empty space. Otherwise, you must enter **c** or **w** to specify window crossing selection. (PICKAUTO)
- Change the size of the pickbox. (PICKBOX)
- Select all objects in a group when you select one object in that group.
- Include the boundary in the selection set when you select a hatch.

To change the size of the pickbox cursor

- 1 Click Tools menu ▶ Options ▶ Selection tab.
- 2 Under Pickbox Size, move the slider until the pickbox is the size you want to use.
- 3 Click OK.

Command line: OPTIONS

To change object selection settings

- 1 Click Tools menu ▶ Options.
- 2 In the Options dialog box, Selection tab, make changes to the Selection Preview and Selection Modes areas and the pickbox size.
- 3 Click OK.

Command line: OPTIONS

To turn on or turn off selection previewing

- 1 Click Tools menu ▶ Options.
- 2 In the Options dialog box, Selection tab, select or clear options as follows:
 - Select the When a Command Is Active option to display the check mark.
 - Select the When No Command Is Active option to display the check mark.
 - Select both options to turn on selection preview whenever it is available.
 - Clear both options to turn off selection preview entirely.

Command line: SELECTIONPREVIEW

To change the appearance of selection previewing

- 1 Click Tools menu ▶ Options.
- 2 In the Options dialog box, Selection tab, click Visual Effect Settings.

- 3** In the Visual Effect Settings dialog box, select one of the following options:
 - **Dash.** Displays dashed lines.
 - **Thicken.** Displays thickened lines.
 - **Both.** Displays dashed and thickened lines.
- 4** Click OK to exit each dialog box.

To exclude objects from selection previewing

- 1** Click Tools menu ► Options.
- 2** In the Options dialog box, Selection tab, click Visual Effect Settings.
- 3** In the Visual Effect Settings dialog box, click Advanced Options.
- 4** In the Advanced Preview Options dialog box, select any of the following options to exclude objects from selection previewing:
 - Exclude Objects on Locked Layers
 - Xrefs
 - Tables
 - Groups
 - Multiline Text
 - Hatches
- 5** Click OK to exit each dialog box.

To change the appearance of the selection area

- 1** Click Tools menu ► Options.
- 2** In the Options dialog box, Selection tab, click Visual Effect Settings.
- 3** In the Visual Effect Settings dialog box, change any of the following settings:
 - **Indicate Selection Area.** Select to display effects for selection areas.
 - **Window Selection Color.** Select a color, or click Select Color to display the "Select Color Dialog Box". (WINDOWAREACOLOR system variable)

■ **Crossing Selection Color.** Select a color, or click Select Color to display the Select Color Dialog Box. (CROSSINGAREACOLOR system variable)

■ **Selection Area Opacity.** Use the slider to set transparency for selection areas. The lower the setting, the more transparent the area. A value of 100 makes the area opaque. (SELECTIONAREAOPACITY system variable)

- 4 Click OK to exit each dialog box.

Group Objects

A group is a saved set of objects that you can select and edit together or separately as needed. Groups provide an easy way to combine drawing elements that you need to manipulate as a unit.

See also:

["Create and Use Blocks \(Symbols\)" on page 459](#)

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.

Create Groups

In addition to choosing the objects that will become the members of a group, you can give the group a name and description.

When you create a group, you can give the group a name and description. If you copy a group, the copy is given the default name Ax and is considered unnamed. Unnamed groups are not listed in the Object Grouping dialog box unless you select Include Unnamed.

If you choose a member of a group that can be selected for inclusion in a new group, all members of the former group are included in the new group.

The objects in your drawing can be members of more than one group, and groups themselves can be nested in other groups. You can ungroup a nested group to restore the original group configuration.

Named groups are not maintained when you use a drawing as an external reference or insert it as a block. However, you can bind and then explode the external reference or explode the block to make the group available as an unnamed group.

NOTE Avoid creating large groups containing hundreds or thousands of objects. A large group significantly degrades the performance of this program.

To create a group

- 1 At the Command prompt, enter **group**.
- 2 In the Object Grouping dialog box, under Group Identification, enter a group name and a description.
- 3 In the Create Group area, click New.
The dialog box closes temporarily.
- 4 Select objects and press ENTER.
- 5 Click OK.

Command line: GROUP

Select Objects in Groups

There are several methods for choosing a group, including selecting the group by name or selecting one of the members of the group.

You can select groups by name at the Select Objects prompt. If the PICKSTYLE system variable is set to 1 or 3 and you select any member of a selectable group, all group members that meet the selection criteria are selected. You can also turn group selection on and off by pressing CTRL+ H or SHIFT+CTRL+ A.

All members of selectable groups are also selected when you use object selection cycling (for example, if you want to select an object that lies directly behind

another object). Selecting an object that is a member of more than one selectable group selects all the members of all the groups that contain that object. To select groups for editing with grips, use the pointing device to select the group at the Command prompt.

To control whether all grouped objects can be selected individually

Do one of the following:

- On the command line, enter **pickstyle**. Enter 1 to turn on group selection. Objects within groups can be selected as a group only, not individually.
- On the command line, enter **pickstyle**. Enter 0 to turn off group selection. Objects within groups can be selected individually only, not as a group.
- At any time, turn group selection on and off by pressing either CTRL+H or CTRL+SHIFT+A.

To control selectability for a specific group

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.

In some circumstances, it is useful to control the order in which objects that belong to the same group are selected. For example, a custom routine that generates toolpaths for numerical control devices might depend on a series of contiguous objects in a specified order.

You can reorder group members in two ways: either change the numerical position of individual members or ranges of group members, or reverse the order of all members. The first object in each group is number 0, not number 1.

Change Group Components, Name, or Description

With the Object Grouping dialog box, you can specify objects to be added to or removed from a group at any time. You can also revise a group's name or

description. If deleting an object or removing it from a group leaves the group empty, the group remains defined but without any members.

NOTE Exploding an object such as a block instance or hatch that belongs to a group does not automatically add the resulting components to any group.

Remove Groups

You can delete a group definition by using the “explode” option in the Object Grouping dialog box. This operation is not the same as exploding a block, hatch, or dimension. Objects that belonged to the exploded group remain in the drawing.

As a result, the group is disbanded but the members are not changed in any other way.

To delete a named group

- 1 At the Command prompt, enter **group**.
- 2 In the Object Grouping dialog box, select the group name from the list of groups.
- 3 Under Change Group, select Explode.
- 4 Click OK.
The group is deleted.

Command line: GROUP

To reorder group members

- 1 At the Command prompt, enter **group**.
- 2 In the Object Grouping dialog box, under Change Group, click Re-Order.
- 3 In the Order Group dialog box, under Group Name, select the group to reorder.
- 4 To view the current order of this group, click Highlight.
- 5 In the Object Grouping dialog box, click Next or Previous to view the objects. Click OK when you have finished viewing the order of the objects.
- 6 In the Order Group dialog box, under Remove From Position, enter an object number.
- 7 Under Enter New Position Number for the Object, enter a new position.

- 8 Under Number of Objects, enter the object number or range of numbers to reorder. Click Re-Order.
- 9 Click OK to close each dialog box.

Command line: GROUP

Correct Mistakes

You can backtrack your recent actions using one of several methods.

Undo a Single Action

The simplest method of backtracking is to use Undo on the Standard toolbar or the U command to undo a single action. Many commands include their own U (undo) option so that you can correct mistakes without leaving the command. When you are creating lines and polylines, for example, enter **u** to undo the last segment.

NOTE By default, the UNDOCTL system variable is set to combine consecutive pan and zoom commands into a single operation when you undo or redo.

Undo Several Actions at Once

Use the Mark option of 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 undo actions you've defined as a group.

You can also undo several actions at once with the Undo list on the Standard toolbar.

Reverse the Effect of Undo

You can reverse the effect of a single U or UNDO command by using 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 607.

Cancel a Command

You can cancel a command without completing it by pressing ESC. To change the cancel key assignment, clear the Windows Standard Accelerator Keys option in the Options dialog box, User Preferences tab.

To undo the most recent action

- Click Edit menu ► Undo.

Standard toolbar



Command line: U

To undo a specific number of actions

- 1 On the Standard toolbar, click the Undo list arrow.
A list of actions that you can undo, starting with the most recent action, is displayed.
- 2 Drag to select the actions to undo.
- 3 Click to undo the selected actions.

Standard toolbar



Command line: UNDO

To redo an action

- Click Edit menu ► Redo.

Only the action immediately preceding an UNDO command can be reversed with REDO. You cannot use REDO to repeat another command.

Standard toolbar



Command line: REDO

To redo a specific number of actions

- 1 On the Standard toolbar, click the Redo list arrow.

A list of undo actions that you can redo, starting with the most recent action, is displayed.

- 2 Drag to select the actions to redo.
- 3 Click to redo the selected actions.

Standard toolbar



Command line: MREDO

Erase Objects

You can erase objects from your drawing using several methods.

- Erase them with ERASE.
- Select them and then cut them to the Clipboard with CTRL+X.
- Select them and press DELETE.

You can restore accidentally erased objects using UNDO. The OOPS command restores all objects deleted by the most recent use of ERASE, BLOCK, or WBLOCK.

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.

Remove Unused Definitions and Styles

You can remove unused *named objects*, including block definitions, dimension styles, layers, linetypes, and text styles with PURGE.

See also:

["Correct Mistakes" on page 605](#)

To erase an object

- 1 Click Modify menu ► Erase.
- 2 At the Select Objects prompt, use a selection method to select the objects to be erased or enter an option:
 - Enter **L** (Last) to erase the last object drawn.
 - Enter **p** (Previous) to erase the last selection set.
 - Enter **all** to erase all objects from the drawing.
 - Enter **?** to see a list of all selection methods.
- 3 Press ENTER to end the command.

Modify toolbar



Command line: ERASE

To restore the last erased object

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

The last objects that were removed by ERASE, BLOCK, or WBLOCK are restored.

Command line: OOPS

To cut objects to the Clipboard

- 1 Select the objects you want to cut.
- 2 Click Edit menu ► Cut. You can also press CTRL+X.
The objects are available to be pasted into other Windows applications.

Command line: CUTCLIP

To remove plus-shaped marker blips

- Click View menu ► Redraw.

Command line: REDRAW

To purge an unused linetype

- 1 Click File menu ▶ Drawing Utilities ▶ Purge.

The Purge dialog box displays a tree view of object types with items that can be purged.

- 2 To purge unreferenced linetypes, use one of the following methods:

- To purge all unreferenced linetypes, select Linetypes.
- To purge specific linetypes, double-click Linetypes to expand the tree view. Then select the linetypes to be purged.

If the item you want to purge is not listed, select View Items You Cannot Purge.

- 3 You are prompted to confirm each item in the list. If you do not want to confirm each purge, clear the Confirm Each Item to Be Purged option.

- 4 Click Purge.

To confirm the purging of each item, respond to the prompt by choosing Yes or No, or Yes to All if more than one item is selected.

- 5 Click Close.

Command line: PURGE

Use Windows Cut, Copy, and Paste

When you want to use objects from a drawing file in another application, you can cut or copy these objects to the Clipboard and then paste them from the Clipboard into the other application.

Cut Objects

Cutting deletes selected objects from the drawing and stores them on the Clipboard. The objects are now available to be pasted into other Microsoft® Windows® documents.

Copy Objects

You can use the Clipboard to copy part or all of a drawing into a document created by another application. The objects are copied in vector format, which retains the high resolution in other applications. These objects are stored in WMF (Windows metafile) format in the Clipboard. The information stored in the Clipboard can then be embedded in the other document. Updating the original drawing does not update the copy embedded in the other application.

Paste Objects

Applications use different internal formats to store Clipboard information. When you copy objects to the Clipboard, information is stored in all available formats. When you paste the Clipboard contents into a drawing, the format that retains the most information is used. However, you can override this setting and convert pasted information to AutoCAD format.

Because it is the easiest format to edit, the AutoCAD format is the preferred format for copying objects to and from AutoCAD. It retains all relevant object information, including block references and 3D aspects.

The Windows metafile (picture) format contains screen vector information, and files can be scaled and printed without losing resolution. Use this format to paste objects into Windows applications that support WMF files. Metafiles pasted into AutoCAD are of higher resolution than bitmapped images (BMP files) but are not as easily manipulated as AutoCAD objects. Bitmapped images are raster images consisting of a pattern of pixels and are commonly used by paint applications.

The color of the object doesn't change when copied to the Clipboard. For example, white objects pasted onto a white background won't be visible. Use the WMFBKGND and WMFFOREGND system variables to control whether the background or foreground is transparent for metafile objects pasted into other applications.

You can insert a linked or embedded object from the Clipboard into a drawing with PASTESPEC. If you convert pasted information to AutoCAD format, the object is inserted as a block reference. To edit the pasted information, explode the block reference into its component objects. When you convert a Windows metafile stored on the Clipboard to AutoCAD format, you may lose some scaling precision. To retain proper scaling, save objects in the original drawing as a block (WBLOCK), and then insert them into AutoCAD using INSERT.

To cut objects to the Clipboard

- 1 Select the objects you want to cut.
- 2 Click Edit menu ► Cut. You can also press CTRL+X.
The objects are available to be pasted into other Windows applications.

Command line: CUTCLIP

To copy objects to the Clipboard

- 1 Select the objects you want to copy.
- 2 Click Edit menu ► Copy. You can also press CTRL+C.

Command line: COPYCLIP

To paste objects from the Clipboard

- Click Edit menu ► Paste. You can also press CTRL+V.
The objects currently on the Clipboard are pasted into the drawing.

Command line: PASTECLIP

To convert pasted information to drawing file format

- 1 Click Edit menu ► Paste Special.
- 2 In the Paste Special dialog box, select Paste.
- 3 From the list of formats, select Picture.
- 4 Click OK.

Command line: PASTESPEC

Modify Objects

You can easily modify the size, shape, and location of objects.

See also:

- ["Work with Custom and Proxy Objects"](#)
- [Modify Existing Dimensions](#)

Choose a Method to Modify Objects

You can easily modify the size, shape, and location of objects. You can

- Enter a command first, and then select the objects to modify.
- Select the objects first, and then enter a command to modify them.
- Select and right-click an object to display a shortcut menu with relevant options.
- Double-click an object to display the Properties palette or, in some cases, a dialog box or editor that is specific to that type of object. (You can specify the double click action for each object type by customizing a CUI file and loading it into the program.)

See also:

[Change Text](#)

["Select Objects" on page 588](#)

[Modify Existing Dimensions](#)

["Display and Change the Properties of Objects" on page 273](#)

["Work with Custom and Proxy Objects"](#)

["Create a Double Click Action" in the *Customization Guide*](#)

Move or Rotate Objects

You can move objects to a different location, or change the orientation of objects by rotating them by an angle or to other objects.

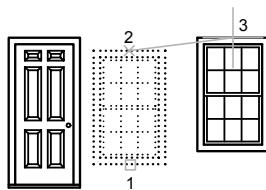
Move Objects

You can move objects at a specified distance and direction from the originals.

Use coordinates, grid snap, object snaps, and other tools to move objects with precision.

Specify Distance with Two Points

Move an object using the distance and direction specified by a base point followed by a second point. In this example, you move the block representing a window. After clicking Move on the Edit menu, select the object to be moved (1). Specify the base point for the move (2) followed by a second point (3). The object is moved the distance and direction of point 2 to point 3.



Specify Distance with Relative Coordinates

You can move an object using a relative distance by entering coordinate values for the first point and pressing ENTER for the second point. The coordinate values are used as a relative displacement rather than the location of a base point.

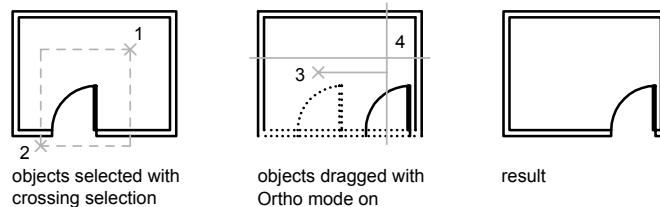
NOTE Do not include an @ sign as you normally would for relative coordinates, because relative coordinates are expected.

To copy objects a specified distance, you can also use direct distance entry with Ortho mode and polar tracking. For more information, see “Enter Direct Distances” on page 387

Use a Stretch-Move

You can also use STRETCH to move objects if all their endpoints lie entirely within the selection window. Turn on Ortho mode or polar tracking to move the objects at a specific angle.

A practical example is moving a door in a wall. The door in the illustration is entirely within a crossing selection, while the wall lines are only partly within the crossing selection area.



The result is that only the endpoints that lie within the crossing selection move.

Use Alternate Methods

You can use grips to move and copy objects quickly. See Use Grips to Edit Objects.

You can also select objects and drag them to a new location; press CTRL to make a copy. Using this method, you can drag objects between open drawings and other applications. If you drag with the right mouse button instead of the left, a shortcut menu is displayed. The menu options include Move Here, Copy Here, Paste as Block, and Cancel. See Embed OLE Objects in Drawings.

To move an object using two points

- 1 Click Modify menu ► Move.
- 2 Select the objects to move.
- 3 Specify a base point for the move.

4 Specify a second point.

The objects you selected are moved to a new location determined by the distance and direction between the first and second points.

Modify toolbar



Command line: MOVE

To move an object using a displacement

- 1** Click Modify menu ► Move.
- 2** Select the object to move.
- 3** Enter the displacement in the form of a Cartesian, polar, cylindrical, or spherical coordinate value. Do not include the @ sign, because a relative coordinate is assumed.
- 4** At the prompt for the second point, press ENTER.

The coordinate values are used as a relative displacement rather than the location of a base point. The selected objects are moved to a new location determined by the relative coordinate values you enter.

Modify toolbar



Command line: MOVE

To move an object from model space to paper space (or vice versa)

- 1** Click a layout tab.
- 2** Click Modify menu ► Change Space.
- 3** Select one or more objects to move.
- 4** Press ENTER.

The object is moved to the new space, and is scaled appropriately to the new space.

Modify toolbar

Command line: CHSPACE

To move by stretching

- 1 Click Modify menu ► Stretch.
- 2 Select the object by using crossing selection.

The crossing selection must include at least one vertex or endpoint. Specify crossing selection by clicking, moving your pointing device from right to left, and clicking again.

- 3 Do *one* of the following:
 - Specify the base point for the move, and then specify a second point.
 - Enter the displacement in the form of a Cartesian, polar, cylindrical, or spherical coordinate value. Do not include the @ sign, because a relative coordinate is assumed. At the prompt for the second point of displacement, press ENTER.

Any objects with at least one vertex or endpoint included within the crossing selection are stretched. Any objects that are completely within the crossing selection are moved without being stretched.

Modify toolbar



Command line: STRETCH

Rotate Objects

You can rotate objects in your drawing around a specified base point.

To determine the angle of rotation, you can enter an angle value, drag using the cursor, or specify a reference angle to align to an absolute angle.

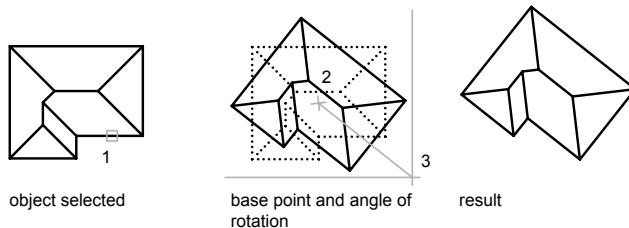
Rotate an Object by a Specified Angle

Enter a rotation angle value from 0 to 360 degrees. You can also enter values in radians, grads, or surveyor bearings. Entering a positive angle value rotates the objects counterclockwise or clockwise, depending on the Direction Control setting in the Drawing Units dialog box.

Rotate an Object by Dragging

Drag the object around the base point and specify a second point. Use Ortho mode, polar tracking, or object snaps for greater precision.

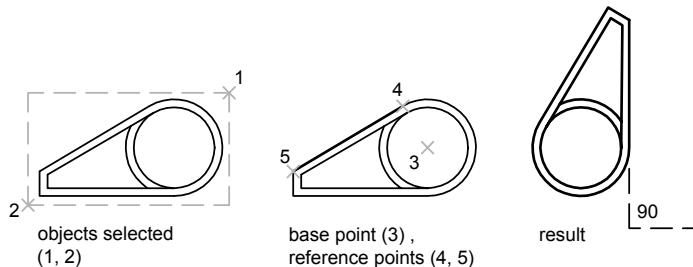
For example, you can rotate the plan view of a house by selecting the objects (1), specifying a base point (2), and specifying an angle of rotation by dragging to another point (3).



Rotate an Object to an Absolute Angle

With the Reference option, you can rotate an object to align it to an absolute angle.

For example, to rotate the part in the illustration so the diagonal edge rotates to 90 degrees, you select the objects to be rotated (1, 2), specify the base point (3), and enter the Reference option. For the reference angle, specify the two endpoints of the diagonal line (4, 5). For the new angle, enter 90.



Rotate an Object in 3D

To rotate 3D objects, you can use either ROTATE or ROTATE3D.

- With ROTATE, you can rotate objects around a specified base point. The axis of rotation passes through the base point and is parallel to the Z axis of the current UCS.

- With ROTATE3D, you can specify the axis of rotation using either two points; an object; the X, Y, or Z axis; or the Z direction of the current view.

See also:

[“Rotate Views in Layout Viewports” on page 235](#)

To rotate an object

- Click Modify menu ► Rotate.
- Select the object to rotate.
- Specify the base point for the rotation.
- Do *one* of the following:
 - Enter the angle of rotation.
 - Drag the object around its base point and specify a point location to which you want to rotate the object.
 - Enter **c** to create a copy of the selected objects.
 - Enter **r** to rotate the selected objects from a specified reference angle to an absolute angle.

Modify toolbar



Command line: ROTATE

To rotate an object to an absolute angle

- Click Modify menu ► Rotate.
- Select the objects to rotate.
- Specify the base point for the rotation.
- Enter **r** (Reference).
- Enter a reference angle value or specify two point locations.
This determines an imaginary line that will be rotated to a new angle.
- Enter the new angle, or specify a point.
The value that you enter for the new angle is an absolute angle, not a relative value. Alternatively, if you specify a point, the reference angle will be rotated to that point.

Modify toolbar



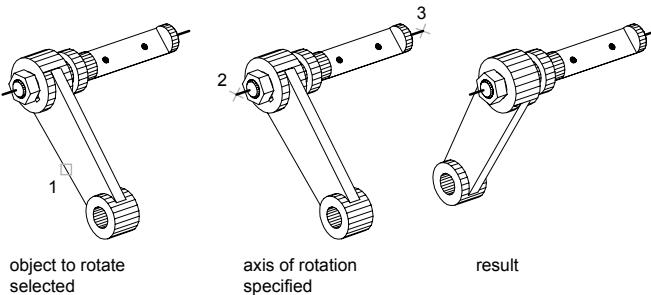
Command line: ROTATE

To rotate a 3D object around an axis

- 1 Click Modify menu > 3D Operations > Rotate 3D.
- 2 Select the object to rotate (1).
- 3 Specify the start point and endpoint of the axis about which the objects are to be rotated (2 and 3).

The positive axis direction is from the start point to the end point, and the rotation follows the right-hand rule (see “Understand the User Coordinate System in 3D” on page 343).

- 4 Specify the angle of rotation.

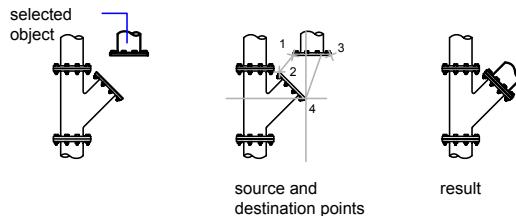


Command line: ROTATE3D

Align Objects

You can move, rotate, or tilt an object so that it aligns with another object.

In the following example, two pairs of points are used to align the piping in 2D using the ALIGN command. Endpoint object snaps align the pipes precisely.



In 3D, use the 3DALIGN command to specify up to three points to define the source plane followed by up to three points to define the destination plane.

- The first source point on an object, called the *base point*, is always moved to the first destination point.
- Specifying a second point for either the source or the destination results in the selected objects being rotated.
- A third point for either the source or the destination results in further rotation of the selected objects.

TIP With 3D solid models, it is recommended that you turn on dynamic UCS to speed the selection of the destination plane.

To align two objects in 2D

- 1 Click Modify menu ► 3D Operations ► Align.
- 2 Select the objects that you want to align.
- 3 Specify a source point and then the corresponding destination point. To rotate the object, specify a second source point followed by a second destination point.
- 4 Press ENTER to end the command.

The selected objects are moved from the source point to the destination point, and second and third points, if you specify them, rotate, and tilt the selected objects.

Command line: ALIGN

To align two objects in 3D

- 1 Click Modify menu ► 3D Operations ► 3D Align.

- 2 Select the objects that you want to align.
- 3 Specify either one, two, or three source points and then the corresponding first, second, or third destination points. The first point is called the *base point*.

The selected objects are moved from the source point to the destination point, and second and third points, if you specify them, rotate, and tilt the selected objects.

Command line: 3DALIGN

Copy, Offset, or Mirror Objects

You can create duplicates of objects in your drawing that are either identical or similar to selected objects.

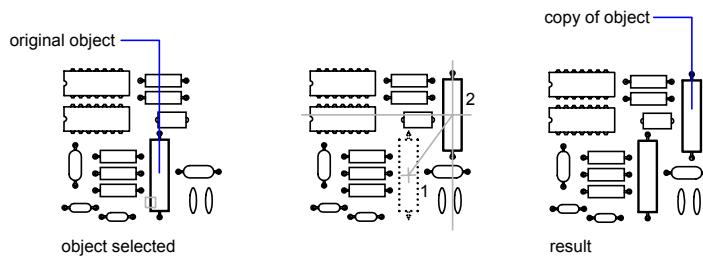
Copy Objects

You can create duplicates of objects at a specified distance and direction from the originals.

Use coordinates, grid snap, object snaps, and other tools to copy objects with precision.

Specify Distance with Two Points

Copy an object using the distance and direction specified by a base point followed by a second point. In this example, you copy the block representing an electronic component. After clicking Copy on the Edit menu, select the original object to be copied. Specify the base point for the move (1) followed by a second point (2). The object is copied the distance and direction of point 1 to point 2.



Specify Distance with Relative Coordinates

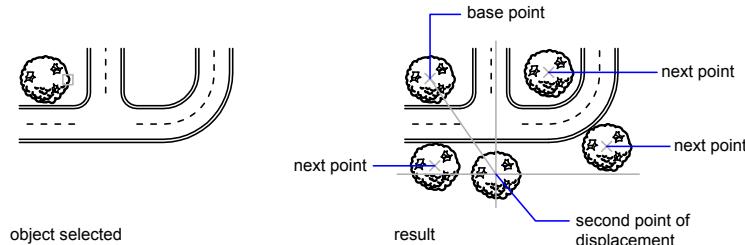
Copy an object using a relative distance by entering coordinate values for the first point and pressing ENTER for the second point. The coordinate values are used as a relative displacement rather than the location of a base point.

NOTE Do not include an @ sign as you normally would for relative coordinates, because relative coordinates are expected.

To copy objects a specified distance, you can also use direct distance entry with Ortho mode and polar tracking. For more information, see “Enter Direct Distances” on page 387.

Create Multiple Copies

The COPY command repeats for convenience. To exit the command, press ENTER.



Use Alternate Methods

You can use grips to move and copy objects quickly. See Use Grips to Edit Objects.

You can also select objects and drag them to a new location; press CTRL to make a copy. Using this method, you can drag objects between open drawings and other applications. If you drag with the right mouse button instead of the left, a shortcut menu is displayed. The menu options include Move Here, Copy Here, Paste as Block, and Cancel. See Embed OLE Objects in Drawings.

To copy an object using two points

- 1 Click Modify menu ► Copy.
- 2 Select the objects to copy.
- 3 Specify the base point.

4 Specify the second point. Press ENTER.

Modify toolbar

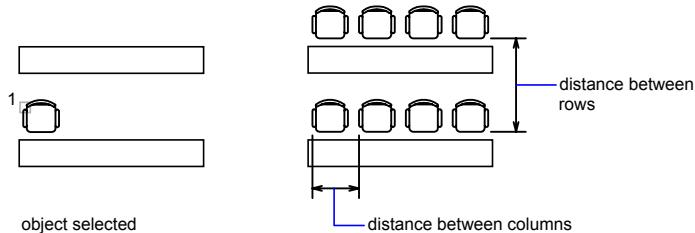


Command line: COPY

Create an Array of Objects

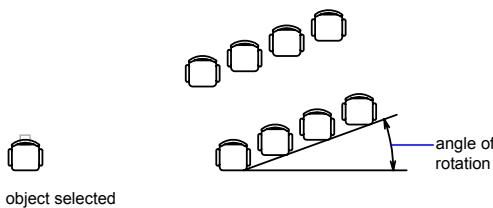
You can create copies of objects in a rectangular or polar (circular) pattern called an array.

For rectangular arrays, you control the number of rows and columns and the distance between each. For polar arrays, you control the number of copies of the object and whether the copies are rotated. To create many regularly spaced objects, arraying is faster than copying.



Create Rectangular Arrays

A rectangular array is built along a baseline defined by the current snap rotation angle. This angle is zero by default, so the rows and columns of a rectangular array are orthogonal with respect to the X and Y axes. The default angle 0 direction setting can be changed in UNITS.



Create Polar Arrays

When you create a polar array, the array is drawn counterclockwise or clockwise, depending on whether you enter a positive or a negative value for the angle to fill.



The radius of the array is determined by the distance from the specified center point to a *reference* or base point on the last selected object. You can use the default reference point (usually an arbitrary point that coincides with a snap point), or you can specify a new base point to be used as the reference point.

Array in 3D

With 3DARRAY, you can create a rectangular array or a polar array of objects in 3D. In addition to specifying the number of columns (X direction) and rows (Y direction), you also specify the number of levels (Z direction).

Limit the Size of Arrays

If you specify a very large number of rows and columns for an array, it may take a long time to create the copies. By default, the number of array elements that can be generated by one command is limited to approximately 100,000. This limit is controlled by the MaxArray setting in the registry.

You can change the limit by setting the MaxArray system registry variable using (**setenv "MaxArray" "n"**) where *n* is a number between 100 and 10000000 (ten million).

NOTE When changing the value of MaxArray, you must enter MaxArray with the capitalization shown.

To create a rectangular array

- 1 Click Modify menu ► Array.
- 2 In the Array dialog box, select Rectangular Array.
- 3 Click Select Objects.

The Array dialog box closes. You are prompted for object selection.

- 4** Select the objects to be arrayed and press ENTER.
- 5** In the Rows and Columns boxes, enter the number of rows and columns in the array.
- 6** Specify the horizontal and vertical spacing (offsets) between objects by using one of the following methods:
 - In the Row Offset and Column Offset boxes, enter the distance between rows and between columns. Adding a plus sign (+) or a minus sign (-) determines direction.
 - Click the Pick Both Offsets button to use the pointing device to specify the diagonal corners of a cell in the array. The cell determines the vertical and horizontal spacing of the rows and columns.
 - Click the Pick Row Offset or Pick Column Offset button to use the pointing device to specify the horizontal and vertical spacing.

The example box displays the result.

- 7** To change the rotation angle of the array, enter the new angle next to Angle of Array.
- 8** The default angle 0 direction setting can also be changed in UNITS.
- 9** Click OK to create the array.

Modify toolbar



Command line: ARRAY

To create a polar array

- 1** Click Modify menu ► Array.
- 2** In the Array dialog box, select Polar Array.
- 3** Next to Center Point, do one of the following:
 - Enter an X value and a Y value for the center point of the polar array.
 - Click the Pick Center Point button. The Array dialog box closes and you are prompted for object selection. Use the pointing device to specify the center point of the polar array.
- 4** Click Select Objects.

The Array dialog box closes and you are prompted for object selection.

- 5** Select the objects to be arrayed.
 - 6** In the Method box, select one of the following methods:
 - Total Number of Items & Angle to Fill
 - Total Number of Items & Angle Between Items
 - Angle to Fill & Angle Between Items
 - 7** Enter the number of items (including the original object), if available.
 - 8** Use one of the following methods:
 - Enter the angle to fill and angle between items, if available. Angle to Fill specifies the distance to fill around the circumference of the array. Angle Between Items specifies the distance between each item.
 - Click the Pick Angle to Fill button and the Pick Angle Between Items button. Use the pointing device to specify the angle to fill and the angle between items.
- The example box displays the result.
- 9** You can set any of the following options:
 - To rotate the objects as they are arrayed, select Rotate Items As Copied. The example area displays the result.
 - To specify the X,Y base point, select More, clear the Set to Object's Default option and enter values in the X and Y boxes, or click the Pick Base Point button and use the pointing device to specify the point.
 - 10** Click OK to create the array.

Modify toolbar

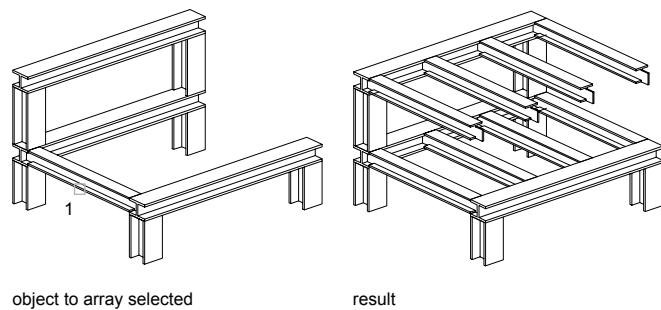


Command line: ARRAY

To create a 3D rectangular array of objects

- 1** Click Modify menu ► 3D Operations ► 3D Array.
- 2** Select the object to array (1).
- 3** Specify Rectangular.
- 4** Enter the number of rows.

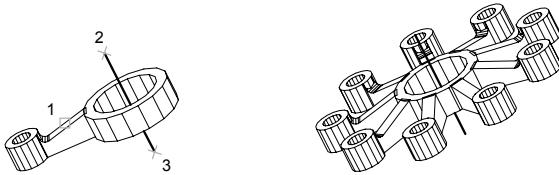
- 5** Enter the number of columns.
- 6** Enter the number of levels.
- 7** Specify the distance between rows.
- 8** Specify the distance between columns.
- 9** Specify the distance between levels.



Command line: 3DARRAY

To create a 3D polar array of objects

- 1** Click Modify menu ► 3D Operations ► 3D Array.
- 2** Select the object to array (1).
- 3** Specify Polar.
- 4** Enter the number of items to array.
- 5** Specify the angle that the arrayed objects are to fill.
- 6** Press ENTER to rotate the objects as they are arrayed, or enter **n** to retain their orientation.
- 7** Specify the start point and endpoint of the axis about which the objects are to be rotated (2 and 3).



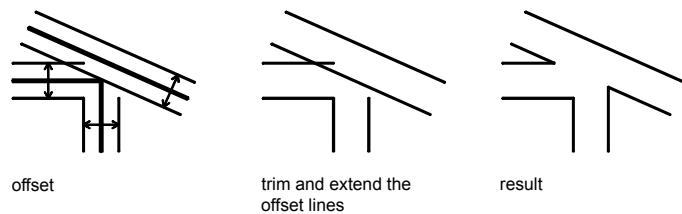
Command line: 3DARRAY

Offset an Object

OFFSET creates a new object whose shape parallels the shape of a selected object. Offsetting a circle or an arc creates a larger or smaller circle or arc, depending on which side you specify for the offset.



A highly effective drawing technique is to offset objects and then trim or extend their ends.



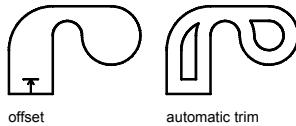
You can offset

- Lines

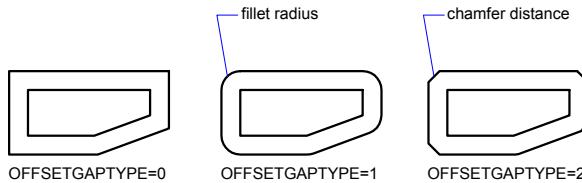
- Arcs
- Circles
- Ellipses and elliptical arcs (resulting in an oval-shaped spline)
- 2D polylines
- Construction lines (xlines) and rays
- Splines

Special Cases for Offset Polylines and Splines

2D polylines and splines are trimmed automatically when the offset distance is larger than can otherwise be accommodated.



Closed 2D polylines that are offset to create larger polylines result in potential gaps between segments. The `OFFSETGAPTYPE` system variable controls how these potential gaps are closed.



To offset an object by specifying a distance

- 1 Click Modify menu > Offset.
- 2 Specify the offset distance.
You can enter a value or use the pointing device.
- 3 Select the object to offset.
- 4 Specify a point on the side where you want to place the new objects.

5 Select another object to offset, or press ENTER to end the command.

Modify toolbar



Command line: OFFSET

To offset an object through a point

- 1** Click Modify menu ► Offset.
- 2** Enter **t** (Through).
- 3** Select the object to offset.
- 4** Specify the through point.
- 5** Select another object to offset, or press ENTER to end the command.

Modify toolbar



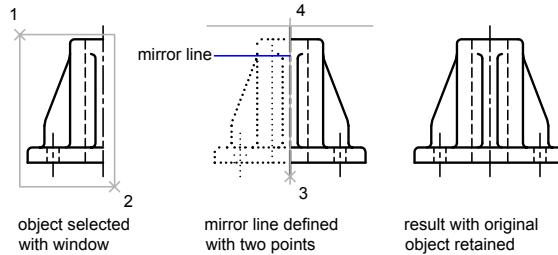
Command line: OFFSET

Mirror Objects

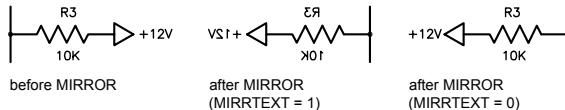
You can flip objects about a specified axis to create a symmetrical mirror image.

Mirroring is useful for creating symmetrical objects because you can quickly draw half the object and then mirror it instead of drawing the entire object.

You flip objects about an axis called a mirror line to create a mirror image. To specify this temporary mirror line, you enter two points. You can choose whether to erase or retain the original objects.



By default, when you mirror text, attributes, and attribute definitions, they are not reversed or turned upside down in the mirror image. The text has the same alignment and justification as before the object was mirrored. If you do want text to be reversed, set the MIRRTEXT system variable to 1.

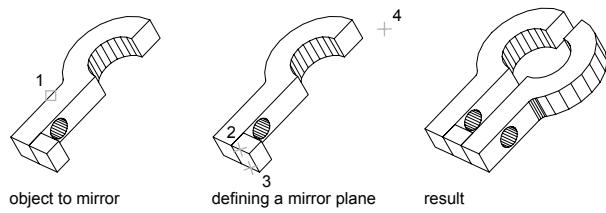


MIRRTEXT affects text that is created with the TEXT, ATTDEF, or MTEXT commands; attribute definitions; and variable attributes. Text and constant attributes that are part of an inserted block are reversed when the block is mirrored regardless of the MIRRTEXT setting.

Mirror in 3D

With MIRROR3D, you can mirror objects across a specified mirroring plane. The mirroring plane can be one of the following:

- The plane of a planar object
- A plane parallel to the XY, YZ, or XZ plane of the current UCS that passes through a specified point
- A plane defined by three specified points (2, 3, and 4)



To mirror objects

- 1 Click Modify menu ► Mirror.
- 2 Select the objects to mirror.
- 3 Specify the first point of the mirror line.
- 4 Specify the second point.
- 5 Press ENTER to retain the original objects, or enter **y** to erase them.

Modify toolbar



Command line: MIRROR

To mirror objects in 3D

- 1 Click Modify menu ► 3D Operations ► Mirror 3D.
- 2 Select the object to mirror.
- 3 Specify three points to define a mirroring plane.
- 4 Press ENTER to retain the original objects, or enter **y** to delete them.

Command line: MIRROR3D

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.

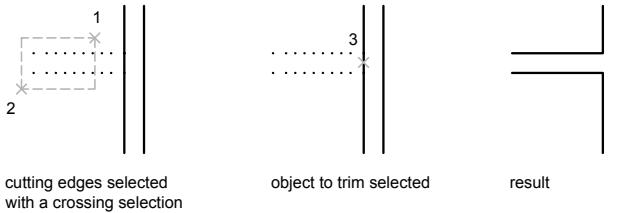
If you do not specify a boundary and press ENTER at the Select Objects prompt, all displayed objects become potential boundaries.

NOTE To select cutting edges or boundary edges that include blocks, you can use only the single selection, Crossing, Fence, and Select All options.

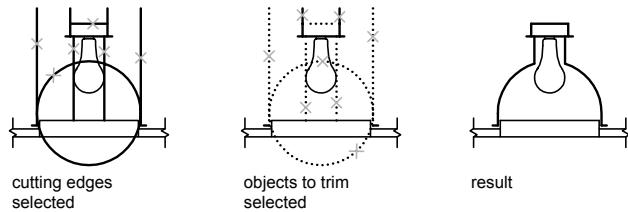
Trim Objects

You can trim objects so that they end precisely at boundary edges defined by other objects.

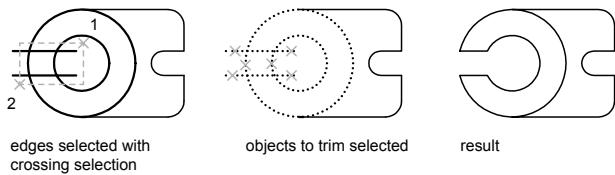
For example, you can clean up the intersection of two walls smoothly by trimming.



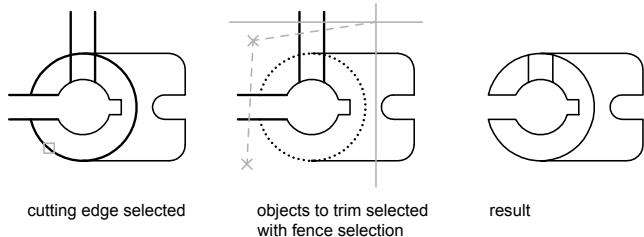
An object can be one of the cutting edges and one of the objects being trimmed. For example, in the illustrated light fixture, the circle is a cutting edge for the construction lines and is also being trimmed.



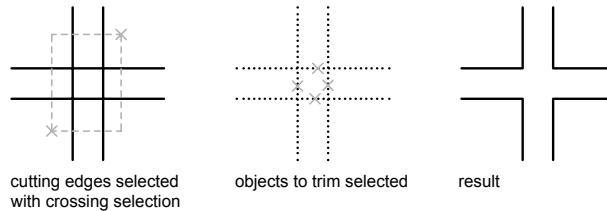
When you trim several objects, the different selection methods can help you choose the current cutting edges and objects to trim. In the following example, the cutting edges are selected using crossing selection.



The following example uses the fence selection method to select a series of objects for trimming.



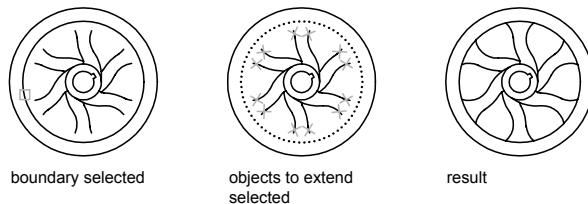
You can trim objects to their nearest intersection with other objects. Instead of selecting cutting edges, you press ENTER. Then, when you select the objects to trim, the nearest displayed objects act as cutting edges. In this example, the walls are trimmed so that they intersect smoothly.



You can extend objects without leaving the TRIM command. Hold down SHIFT and select the objects to be extended.

Extend Objects

Extending operates the same way as trimming. You can extend objects so they end precisely at boundary edges defined by other objects. In this example, you extend the lines precisely to a circle, which is the boundary edge.

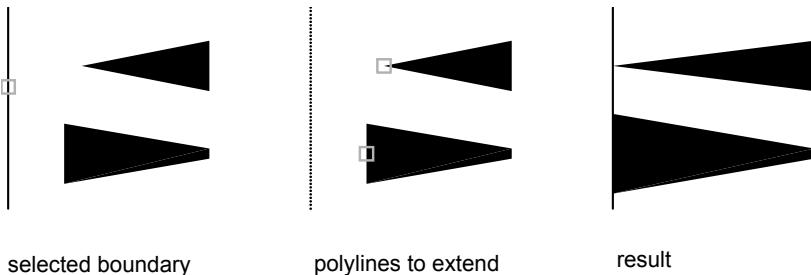


You can trim objects without leaving the EXTEND command. Hold down SHIFT and select the objects to be trimmed.

Trim and Extend Wide Polylines

2D wide polylines trim and extend at their centerlines. The ends of wide polylines are always square. Trimming a wide polyline at an angle causes portions of the end to extend beyond the cutting edge

If you trim or extend a tapered 2D polyline segment, the width of the extended end is corrected to continue the original taper to the new endpoint. If this correction gives the segment a negative ending width, the ending width is forced to 0.



Trim and Extend Spline-Fit Polylines

Trimming a spline-fit polyline removes the curve-fit information and changes the spline-fit segments into ordinary polyline segments.

Extending a spline-fit polyline adds a new vertex to the control frame for the polyline.

Trim or Extend in 3D

You can trim or extend an object to any other object in 3D space, regardless of whether the objects are on the same plane or parallel to the cutting or boundary edges. In the TRIM and EXTEND commands, use the Project and Edge options to select one of three projections for trimming or extending:

- The XY plane of the current UCS
- The plane of the current view
- True 3D, which is not a projection

See also:

[“Break and Join Objects” on page 652](#)

To extend an object

- 1 Click Modify menu ▶ Extend.
- 2 Select the objects to serve as boundary edges.
To select all displayed objects as potential boundary edges, press ENTER without selecting any objects.
- 3 Select the objects to extend.

Modify toolbar



Command line: EXTEND

To trim an object

- 1 Click Modify menu ► Trim.
- 2 Select the objects to serve as cutting edges.
To select all displayed objects as potential cutting edges, press ENTER without selecting any objects.
- 3 Select the objects to trim.

Modify toolbar



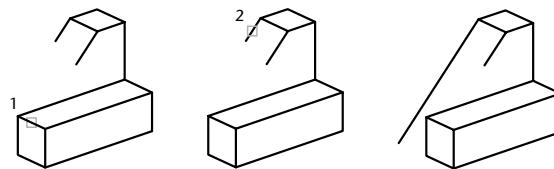
Command line: TRIM

To extend objects in 3D wireframe models

- 1 Click Modify menu ► Extend.



- 2 Select the boundary edge for extending (1).
- 3 Enter **e** (Edge).
- 4 Enter **e** (Extend).
- 5 Enter **p** (Project).
- 6 Enter **u** (UCS).
- 7 Select the object to extend (2).



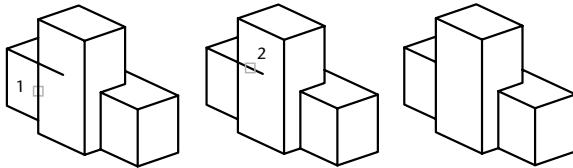
Command line: EXTEND

To trim in 3D using the current view plane

- 1 Click Modify menu ► Trim.



- 2 Select the cutting edge for trimming (1).
- 3 Enter **p** (Project).
- 4 Enter **v** (View).
- 5 Select the object to trim (2).



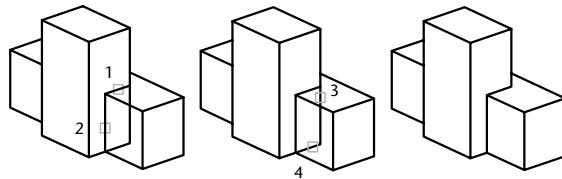
Command line: TRIM

To trim objects in 3D wireframe models

- 1 Click Modify menu ► Trim.



- 2 Select the cutting edges to use for trimming (1 and 2).
- 3 Enter **p** (Project).
- 4 Enter **n** (None).
- 5 Select the object to trim (3 and 4).



Command line: TRIM

Resize or Reshape Objects

You can resize objects to make them longer or shorter in only one direction or to make them proportionally larger or smaller.

You can also stretch certain objects by moving an endpoint, vertex, or control point.

Lengthen Objects

With LENGTHEN, you can change the included angle of arcs and the length of the following objects:

- Lines
- Arcs
- Open polylines
- Elliptical arcs
- Open splines.

The results are similar to extending and trimming. You can

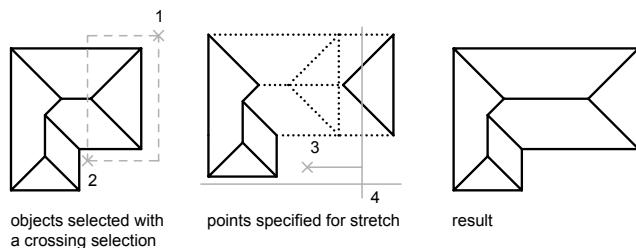
- Drag an object endpoint dynamically
- Specify a new length or angle as a percentage of the total length or angle
- Specify an incremental length or angle measured from an endpoint
- Specify the object's total absolute length or included angle

Stretch Objects

With STRETCH, you relocate the endpoints of objects that lie across or within a crossing selection window.

- Objects that are partially enclosed by a crossing window are stretched.
- Objects that are completely enclosed within the crossing window, or that are selected individually, are moved rather than stretched.

To stretch an object, you specify a base point and then a point of displacement.



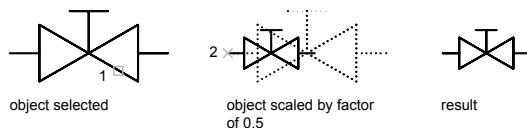
To stretch with precision, use object snaps, grid snaps, and relative coordinate entry.

Scale Objects Using a Scale Factor

With SCALE, you can make an object uniformly larger or smaller. To scale an object, you specify a base point and a scale factor. Alternatively, you can specify a length to be used as a scale factor based on the current drawing units.

A scale factor greater than 1 enlarges the object. A scale factor between 0 and 1 shrinks the object.

Scaling changes the size of all dimensions of the selected object. A scale factor greater than 1 enlarges the object. A scale factor less than 1 shrinks the object.



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 652](#)

To stretch an object

- 1 Click Modify menu ► Stretch.
- 2 Select the object using a crossing window selection.
The crossing window must include at least one vertex or endpoint.
- 3 Do *one* of the following:
 - Enter the displacement in the form of a relative Cartesian, polar, cylindrical, or spherical coordinate. Do not include the @ sign, because a relative coordinate is assumed. Press ENTER at the prompt for the second point of displacement.
 - Specify the base point for the stretch, and then specify a second point, to determine the distance and direction.

Any objects with at least one vertex or endpoint included within the crossing window are stretched. Any objects that are completely within the crossing window, or selected individually, are moved without stretching.

Modify toolbar



Command line: STRETCH

To scale an object by a scale factor

- 1 Click Modify menu ► Scale.
- 2 Select the object to scale.

- 3** Specify the base point.
- 4** Enter the scale factor or drag and click to specify a new scale.

Modify toolbar



Command line: SCALE

To scale an object by reference

- 1** Click Modify menu ► Scale.
- 2** Select the object to scale.
- 3** Select the base point.
- 4** Enter **r** (Reference).
- 5** Select the first and second reference points, or enter a value for the reference length.

Modify toolbar



Command line: SCALE

To change the length of an object by dragging

- 1** Click Modify menu ► Lengthen.
- 2** Enter **dy** (Dynamic Dragging mode).
- 3** Select the object you want to lengthen.
- 4** Drag the endpoint closest to the point of selection, and specify a new endpoint.

The selected object is lengthened or shortened without changing its location or orientation.

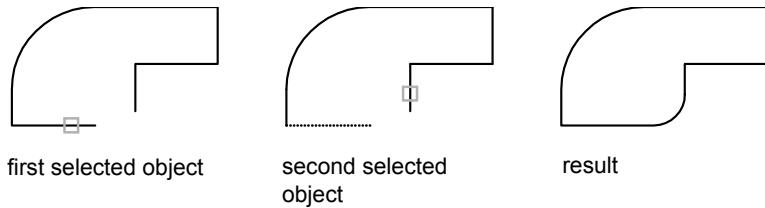
Command line: LENGTHEN

Fillet, Chamfer, Break, or Join Objects

You can change objects to meet in rounded or flattened corners. You can also create or close gaps in objects.

Create Fillets

A fillet connects two objects with an arc that is tangent to the objects and has a specified radius.



An inside corner is called a fillet and an outside corner is called a round; you can create both using the FILLET command.

You can fillet

- Arcs
- Circles
- Ellipses and elliptical arcs
- Lines
- Polylines
- Rays
- Splines
- Xlines
- 3D solids

FILLET can be used to round all corners on a polyline using a single command.

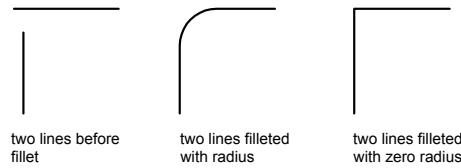
NOTE Filleting a hatch boundary that was defined from line segments removes hatch associativity. If the hatch boundary was defined from a polyline, associativity is maintained.

If both objects being filleted are on the same layer, the fillet arc is created on that layer. Otherwise, the fillet arc is created on the current layer. The layer affects object properties including color and linetype.

Use the Multiple option to fillet more than one set of objects without leaving the command.

Set the Fillet Radius

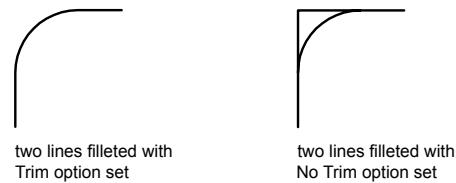
The fillet radius is the radius of the arc that connects filleted objects. Changing the fillet radius affects subsequent fillets. If you set the fillet radius to 0, filleted objects are trimmed or extended until they intersect, but no arc is created.



You can hold down SHIFT while selecting the objects to override the current fillet radius with a value of 0.

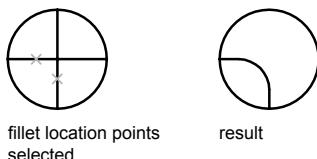
Trim and Extend Filleted Objects

You can use the Trim option to specify whether the selected objects are trimmed or extended to the endpoints of the resulting arc or left unchanged.



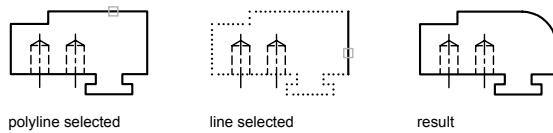
Control the Location of the Fillet

Depending on the locations you specify, more than one possible fillet can exist between the selected objects. Compare the selection locations and resulting fillets in the illustrations.



Fillet Line and Polyline Combinations

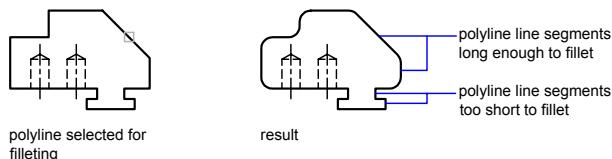
To fillet lines with polylines, each line or its extension must intersect one of the polyline line segments. If the Trim option is on, the filleted objects and the fillet arc join to form a single new polyline.



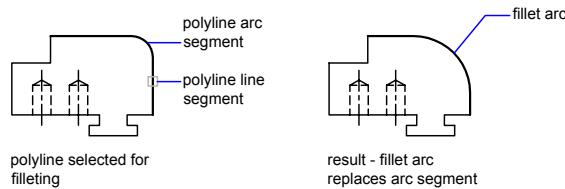
Fillet an Entire Polyline

You can fillet an entire polyline or remove fillets from an entire polyline.

If you set a nonzero fillet radius, FILLET inserts fillet arcs at the vertex of each polyline segment that is long enough to accommodate the fillet radius.



If two polyline line segments converge as they approach an arc segment that separates them, FILLET removes the arc segment and replaces it with a fillet arc.

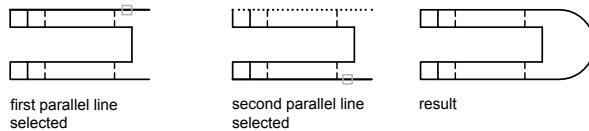


If you set the fillet radius to 0, no fillet arcs are inserted. If two polyline line segments are separated by one arc segment, FILLET removes that arc and extends the lines until they intersect.

Fillet Parallel Lines

You can fillet parallel lines, xlines, and rays. The current fillet radius temporarily adjusts to create an arc that is tangent to both objects and located in the plane common to both objects.

The first selected object must be a line or a ray, but the second object can be a line, an xline, or a ray. The fillet arc connects as shown in the illustration.



Fillet Objects with Non-Zero Thickness in 3D

You can fillet coplanar objects with extrusion directions not parallel to the Z axis of the current UCS. FILLET determines the extrusion direction for the fillet arc in 3D space closest to the direction of the Z axis of the current UCS.

To set the fillet radius

- 1 Click Modify menu > Fillet.
- 2 Enter **r** (Radius).
- 3 Enter the fillet radius

4 Select the objects to fillet.

Modify toolbar



Command line: FILLET

To fillet two line segments

- 1** Click Modify menu ► Fillet.
- 2** Select the first line.
- 3** Select the second line.

Modify toolbar



Command line: FILLET

To fillet without trimming

- 1** Click Modify menu ► Fillet.
- 2** If necessary, enter **t** (Trim). Enter **n** (No Trim).
- 3** Select the objects to fillet.

Modify toolbar



Command line: FILLET

To fillet an entire polyline

- 1** Click Modify menu ► Fillet.
- 2** Enter **p** (Polyline).
- 3** Select the polyline.

Modify toolbar



Command line: FILLET

To fillet multiple sets of objects

- 1 Click Modify menu ► Fillet.
- 2 Enter **m** (Multiple).
The main prompt is displayed.
- 3 Select the first line, or enter an option and complete the prompts for that option. Select the first line.
- 4 Select the second line.
The main prompt is displayed again.
- 5 Select the first line for the next fillet, or press ENTER or ESC to end the command.

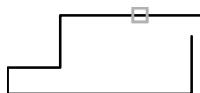
Modify toolbar



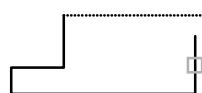
Command line: FILLET

Create Chamfers

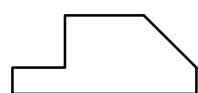
A chamfer connects two objects with an angled line. It is usually used to represent a beveled edge on a corner.



first selected line



second selected line



result

You can chamfer

- Lines
- Polylines
- Rays
- Xlines

■ 3D solids

CHAMFER can be used to bevel all corners of a polyline using a single command.

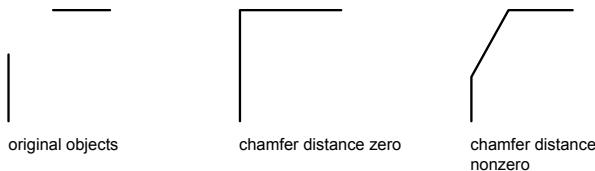
NOTE Chamfering a hatch boundary that was defined from line segments removes hatch associativity. If the hatch boundary was defined from a polyline, associativity is maintained.

If both objects being chamfered are on the same layer, the chamfer line is created on that layer. Otherwise, the chamfer line is created on the current layer. The layer affects object properties including color and linetype.

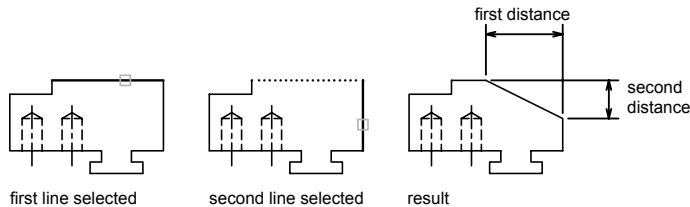
Use the Multiple option to chamfer more than one set of objects without leaving the command.

Chamfer by Specifying Distances

The chamfer distance is the amount each object is trimmed or extended to meet the chamfer line or to intersect the other. If both chamfer distances are 0, chamfering trims or extends the two objects until they intersect but does not create a chamfer line. You can hold down SHIFT while selecting the objects to override the current chamfer distances with a value of 0.



In the following example, you set the chamfer distance to 0.5 for the first line and 0.25 for the second line. After you specify the chamfer distance, you select the two lines as shown.



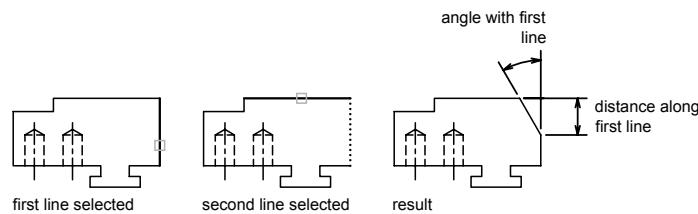
Trim and Extend Chamfered Objects

By default, objects are trimmed when chamfered, but you can use the Trim option to specify that they remain untrimmed.

Chamfer by Specify Length and Angle

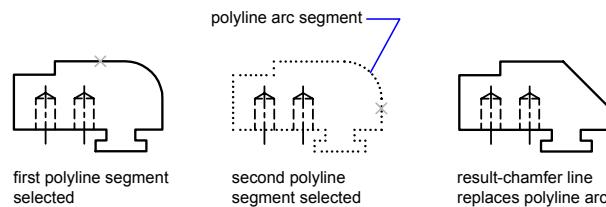
You can chamfer two objects by specifying where on the first selected object the chamfer line starts, and then the angle the chamfer line forms with this object.

In this example, you chamfer two lines so that the chamfer line starts 1.5 units from the intersection along the first line and forms an angle of 30 degrees with this line.



Chamfer Polylines and Polyline Segments

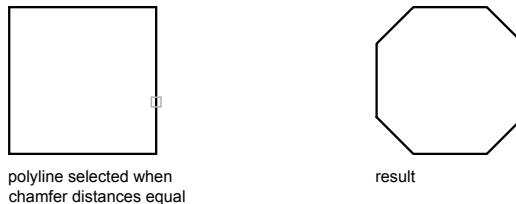
If the two objects you select for chamfering are segments of a polyline, they must be adjacent or separated by no more than one arc segment. If they are separated by an arc segment, as shown in the illustration, chamfering deletes the arc and replaces it with a chamfer line.



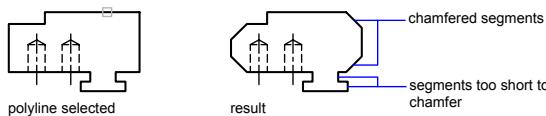
Chamfer an Entire Polyline

When you chamfer an entire polyline, each intersection is chamfered. For best results, keep the first and second chamfer distances equal.

In this example, the chamfer distances are set to equal values.



When you chamfer an entire polyline, only the segments that are long enough to accommodate the chamfer distance are chamfered. The polyline in the following illustration has some segments too short to be chamfered.



To set chamfer distances

- 1 Click Modify menu > Chamfer.
- 2 Enter **d** (Distances).
- 3 Enter the first chamfer distance.
- 4 Enter the second chamfer distance.
- 5 Select the lines for chamfering.

Modify toolbar



Command line: CHAMFER

To chamfer two nonparallel line segments

- 1 Click Modify menu > Chamfer.
- 2 Select the first line.
- 3 Select the second line.

Modify toolbar



Command line: CHAMFER

To chamfer by specifying chamfer length and angle

- 1 Click Modify menu ► Chamfer.
- 2 Enter **a** (Angle).
- 3 Enter the distance from the corner to be chamfered along the first line.
- 4 Enter the chamfer angle.
- 5 Select the first line. Then select the second line.

Modify toolbar



Command line: CHAMFER

To chamfer without trimming

- 1 Click Modify menu ► Chamfer.
- 2 Enter **t** (Trim Control).
- 3 Enter **n** (No Trim).
- 4 Select the objects to chamfer.

Modify toolbar



Command line: CHAMFER

To chamfer an entire polyline

- 1 Click Modify menu ► Chamfer.
- 2 Enter **p** (Polyline).
- 3 Select the polyline.

The polyline is chamfered using the current chamfer method and the default distances.

Modify toolbar



Command line: CHAMFER

To chamfer multiple sets of objects

- 1 Click Modify menu ► Chamfer.
- 2 Enter **m** (Multiple).
The main prompt is displayed.
- 3 Select the first line, or enter an option and complete the prompts for that option and then select the first line.
- 4 Select the second line.
The main prompt is displayed again.
- 5 Select the first line for the next chamfer, or press ENTER or ESC to end the command.

Modify toolbar



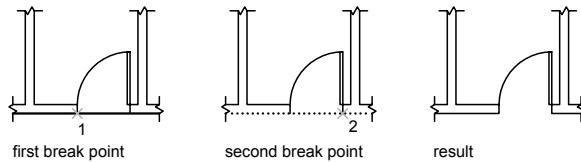
Command line: CHAMFER

Break and Join Objects

You can break an object into two objects with or without a gap between them. You can also join objects to make a single object.

Break Objects

Use BREAK to create a gap in an object, resulting in two objects with a gap between them. BREAK is often used to create space for block or text.



To break an object without creating a gap, specify both break points at the same location. The fastest way to do this is to enter @0,0 at the prompt for the second point.

You can create breaks in most geometric objects except

- Blocks
- Dimensions
- Mlines
- Regions

Join Objects

Use JOIN to combine similar objects into a single object. You can also create complete circles and ellipses from arcs and elliptical arcs. You can join

- Arcs
- Elliptical arcs
- Lines
- Polylines
- Splines

The object to which you want to join similar objects is called a source object. Objects to be joined must be located in the same plane. Additional restrictions for each type of objects are described in the JOIN command.

NOTE When joining two or more arcs (or elliptical arcs), the arcs are joined counterclockwise beginning from the source object.

See also:

- [Modify or Join Polylines](#)
- [Modify Complex Objects](#)

To break an object

1 Click Modify menu ► Break.

2 Select the object to break.

By default, the point at which you select the object is the first break point. To select a different pair of break points, enter **f** (First) and specify the first break point.

3 Specify the second break point.

To break an object without creating a gap, enter **@0,0** to specify the previous point.

Modify toolbar



Command line: BREAK

To join objects

1 Click Modify menu ► Join.

2 Select the source object to which you want to join objects.

3 Select one or more objects to join to the source object.

Valid objects include arcs, elliptical arcs, lines, polylines, and splines. Additional restrictions for each type of objects are described in the JOIN command.

Modify toolbar



Command line: JOIN

Use Grips to Edit Objects

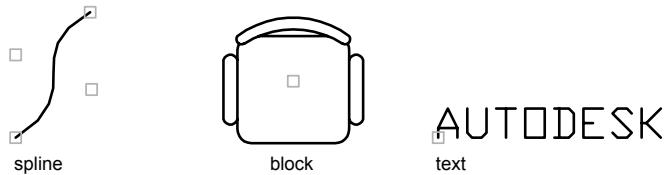
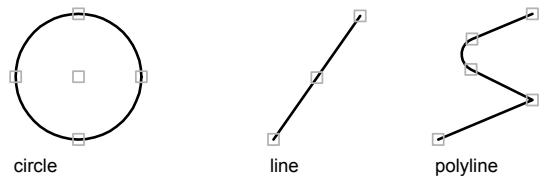
Grips are small squares that are displayed at strategic points on objects that you have selected with a pointing device. You can drag these grips to edit objects directly and quickly.

Use Grip Modes

You can drag grips to perform any stretch, move, rotate, scale, or mirror operations. The editing operation you choose to perform is called a *grip mode*.

Grips are small, solid-filled squares that are displayed at strategic points on objects that you have selected with a pointing device. You can drag these grips to stretch, move, rotate, scale, or mirror objects quickly.

When grips are on, you can select the objects you want to manipulate *before* entering a command, and then you can manipulate the objects with the pointing device.



To use a grip mode, select a grip (base grip) to act as the base point for the action. (A selected grip is also called a *hot grip*.) Then select one of the grip modes. You can cycle through these modes by pressing ENTER or SPACEBAR. You can also use shortcut keys or right-click to see all of the modes and options.

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.

For quadrant grips on circles and ellipses, distance is measured from the center point, not the selected grip. For example, in Stretch mode, you can select a quadrant grip to stretch a circle and then specify a distance on the command line for the new radius. The distance is measured from the center of the circle, not the selected quadrant. If you select the center point to stretch the circle, the circle moves.

When a 2D object lies on a plane other than the current UCS, the object is stretched on the plane on which it was created, not on the plane of the current UCS.

You can limit the display of grips on selected objects. The GRIPOBJLIMIT system variable suppresses the display of grips when the initial selection set includes more than the specified number of objects. If you add objects to the current selection set, the limit does not apply. For example, if GRIPOBJLIMIT is set to 20, you can select 15 objects and then add 25 objects to the selection, and all objects display grips.

Stretch with Grips

You can stretch an object by moving selected grips to new locations. Grips on text, block references, midpoints of lines, centers of circles, and point objects move the object rather than stretching it. This is an excellent method for moving block references and adjusting dimensions.

Move with Grips

You can move objects by the grip selected. Selected objects are highlighted and are moved the direction and distance of the next point location you specify.

Rotate with Grips

You can rotate selected objects around a base point by dragging and specifying a point location. Alternatively, you can enter an angle value. This is an excellent method for rotating block references.

Scale with Grips

You can scale selected objects relative to a base point. Increase the size of an object by dragging outward from the base grip and specifying a point location, or decrease the size by dragging inward. Alternatively, you can enter a value for relative scaling.

Mirror with Grips

You can mirror selected objects across a temporary mirror line. Turning Ortho on helps you specify a vertical or horizontal mirror line.

See also:

Use Dynamic Input

["Work with Dynamic Blocks in Drawings" on page 560](#)

To turn on grips

- 1 On the Tools menu, click Options.
- 2 In the Options dialog box, Selection tab, select Enable Grips.
- 3 Click OK.

Command line: OPTIONS

To set the color of the unselected grip beneath the cursor

- 1 Click Tools menu ► Options.
- 2 In the Options dialog box, Selection tab, click the arrow under Hover Grip Color.
- 3 Select a color, or click Select Color to display the Select Color dialog box.

To limit the number of objects in the initial selection set that display grips

- 1 Click Tools menu ► Options.
- 2 In the Options dialog box, Selection tab, enter a number in the Object Selection Limit for Display of Grips box. The maximum is 32,767.
If you add objects to the current selection set, the limit does not apply.

To display grip tips for custom objects that support grip tips

- 1 Click Tools menu ► Options.
- 2 In the Options dialog box, Selection tab, select Enable Grip Tips.

To cancel grip selection

- Press ESC.

To stretch an object using grips

- 1 Select the object to stretch.
- 2 Select a base grip on the object.
The selected grip is highlighted, and Stretch, the default grip mode, is active.
- 3 Move the pointing device and click.
The selected object is stretched as the grip moves.

NOTE To copy the selected object while you're stretching it, press and hold the CTRL key while you're stretching the object.

To stretch using more than one grip

- 1 Select several objects to stretch.
- 2 Hold down SHIFT and click several grips so that they are highlighted.
- 3 Release SHIFT and select a grip as the base grip by clicking the grip.
The default grip mode, Stretch, is active.
- 4 Move the pointing device and click.
The selected grips act in unison and the selected objects are stretched.

To move objects using grips

- 1 Select the objects to move.
- 2 Select a base grip on an object by clicking the grip.
The selected grip is highlighted, and Stretch, the default grip mode, is active.
- 3 Cycle through the grip modes by pressing ENTER until the grip mode Move appears.
Alternatively, you can right-click to display a shortcut menu of modes and options.
- 4 Move the pointing device and click.
The selected objects are moved along with the grip.

NOTE To copy the selected object while you're moving it, press and hold the CTRL key while you're moving the object.

To rotate objects using grips

- 1 Select the objects to rotate.
- 2 Select a base grip on an object by clicking the grip.
The selected grip is highlighted, and Stretch, the default grip mode, is active.
- 3 Cycle through the grip modes by pressing ENTER until the grip mode Rotate appears.
Alternatively, you can right-click to display shortcut menu modes and options.

- 4 Move the pointing device and click.

The selected objects are rotated around the base grip.

NOTE To copy the selected object while you're rotating it, press and hold the CTRL key while you're rotating the object.

To scale objects using grips

- 1 Select the objects to scale.

- 2 Select a base grip on an object by clicking the grip.

The selected grip is highlighted, and Stretch, the default grip mode, is active.

- 3 Cycle through the grip modes by pressing ENTER until the grip mode Scale appears.

Alternatively, you can right-click to display shortcut menu modes and options.

- 4 Enter the scale factor or drag and click to specify a new scale.

NOTE To copy the selected object while you're scaling it, press and hold the CTRL key while you're scaling the object.

To mirror objects using grips

- 1 Select the objects to mirror.

- 2 Select a base grip on an object by clicking the grip.

The selected grip is highlighted, and the default grip mode, Stretch, is active.

- 3 Cycle through the grip modes by pressing ENTER until the grip mode Mirror appears.

Alternatively, you can right-click to display shortcut menu modes and options.

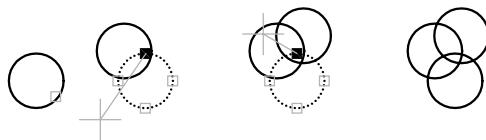
- 4 Click to specify the second point of the mirror line.

Turning on Ortho mode is often useful when mirroring objects.

Make Multiple Copies with Grips

You can create multiple copies of objects as you modify them with any of the grip modes.

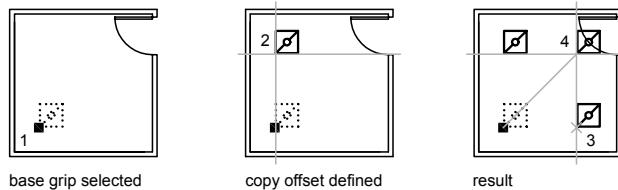
For example, by using the Copy option, you can rotate the selected objects, leaving copies at each location you specify with the pointing device.



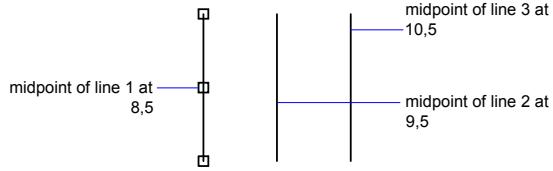
You can also make multiple copies by holding down CTRL as you select the first point. For example, with the Stretch grip mode, you can stretch an object, such as a line, and then copy it to any point in the drawing area. Multiple copies continue being made until you turn off grips.

Define an Offset Snap or a Rotation Snap

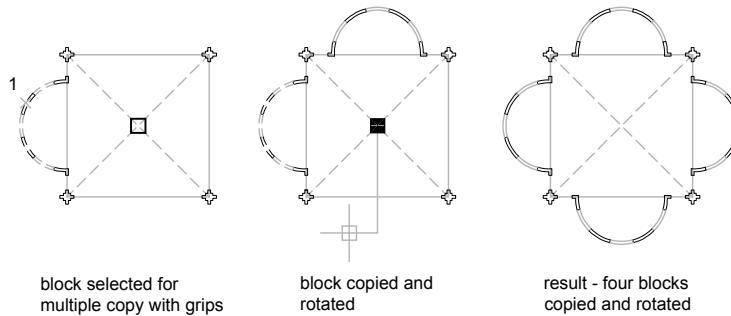
You can place multiple copies at regularly spaced intervals with an offset snap. The offset snap is defined by the distance between an object and the next copy. In the lighting layout below, the first copy of the light fixture symbol is placed at an offset of two units. All subsequent copies are then placed two units apart.



If you hold down CTRL while you select multiple copy points with the pointing device, the graphics cursor snaps to an offset point based on the last two points you selected. In the illustration below, the midpoint of line 1 is at coordinate 8,5. Based on that midpoint, line 2 was copied using the CTRL key and Stretch grip mode; its midpoint is at 9,5. The third line snaps to an offset based on the coordinate values 10,5.



Similarly, you can place multiple copies at angular intervals around a base grip with a rotation snap. The rotation snap is defined as the angle between an object and the next copy when you are using Rotate grip mode. Hold down CTRL to use the rotation snap.



To create copies in any grip mode

- 1 Select the objects to copy.
- 2 Select a base grip on an object by clicking the grip.
The selected grip is highlighted, and the default grip mode, Stretch, is active.
- 3 Cycle through the grip modes by pressing ENTER until the grip mode you want appears.
Alternatively, you can right-click to display shortcut menu modes and options.
- 4 Enter **c** (Copy) or press and hold the CTRL key while you stretch, move, rotate, or scale.
Copies continue being made until you turn off grips.
- 5 Enter or specify the additional input required for the current grip mode.

- 6 Turn off grips by pressing ENTER, SPACEBAR, or ESC.

To create an offset snap for multiple copies using grips

- 1 Select the objects to copy.
- 2 Select a base grip on an object by clicking the grip.
The selected grip is highlighted, and the default grip mode, Stretch, is active.
- 3 Cycle through the grip modes by pressing ENTER until the grip mode Move appears.
Alternatively, you can right-click to display shortcut menu modes and options.
- 4 Enter **c** (Copy).
- 5 Move the cursor and click.
The offset snap is the distance between the grip you selected and the location you specified for the copy.
- 6 Hold down CTRL and place additional copies by specifying additional locations.
These copies are created at the same offset snap distance as the last copy.
- 7 Turn off grips by pressing ENTER, SPACEBAR, or ESC.

To create a rotation snap for multiple rotated copies using grips

- 1 Select the objects to rotate.
- 2 Select a base grip on an object by clicking the grip.
The selected grip is highlighted, and the default grip mode, Stretch, is active.
- 3 Cycle through the grip modes by pressing ENTER until the grip mode Rotate appears.
Alternatively, you can right-click to display shortcut menu modes and options.
- 4 Enter **c** (Copy).
- 5 Move the pointing device and click.
The rotation snap is the angle between the grip you selected and the location you specified for the copy.
- 6 Hold down CTRL and place additional copies by specifying additional locations.

These copies are created at the same rotation snap angle as the first copy.

- 7 Turn off grips by pressing ENTER, SPACEBAR, or ESC.

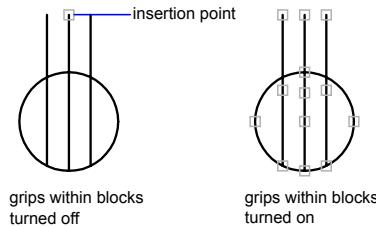
To mirror objects and retain the originals using grips

- 1 Select the objects to mirror.
- 2 Select a base grip on an object by clicking the grip.
The selected grip is highlighted, and the default grip mode, Stretch, is active.
- 3 Cycle through the grip modes by pressing ENTER until the grip mode Mirror appears.
Alternatively, you can right-click to display shortcut menu modes and options.
- 4 Hold down CTRL (or enter **c** for Copy) to retain the original image, and specify the second point of the mirror line.
Turning on Ortho mode is often useful when mirroring objects.
- 5 Turn off grips by pressing ENTER, SPACEBAR, or ESC.

Control Grips in Blocks

You can specify whether a block displays a single grip or multiple grips.

You can specify whether a selected block reference displays a single grip at its insertion point or displays multiple grips associated with the objects grouped within the block.



See also:

[Specify Grips for Dynamic Blocks](#)

To turn grips within blocks on or off

- 1 Click Tools menu ► Options.
- 2 In the Options dialog box, Selection tab, select or clear Enable Grips Within Blocks.
- 3 Click OK.

Command line: OPTIONS

Modify Complex Objects

Additional editing operations are available for complex objects, such as blocks, dimensions, hatches, and polylines.

See also:

- “Create 3D Models”
- “Modify Blocks” on page 573
- “Modify Hatches and Solid-Filled Areas” on page 787
- “Change Text” on page 841
- “Modify Existing Dimensions” on page 925
- “Update Fields” on page 828
- “Create and Modify Tables” on page 856
- “Modify Raster Images and Image Boundaries”

Disassociate Compound Objects (Explode)

You can explode a compound object, such as a polyline, dimension, hatch, or block reference, to convert it into individual elements. For example, exploding a polyline breaks it down to simple lines and arcs. Exploding a block reference or an associative dimension replaces it with copies of the objects that compose the block or dimension.

Explode Dimensions and Hatches

When you explode a dimension or a hatch, all associativity is lost and the dimension or hatch object is replaced by individual objects such as lines, text, points, and 2D solids. To explode dimensions automatically when you create them, set the DIMASSOC system variable to 0.

Explode Polylines

When you explode a polyline, any associated width information is discarded. The resulting lines and arcs follow the polyline's centerline. If you explode a

block that contains a polyline, you need to explode the polyline separately. If you explode a donut, its width becomes 0.

Explode Block References

If you explode a block with attributes, the attribute values are lost, leaving only the attribute definitions. The colors and linetypes of objects in exploded block references can change.

Explode External References

An external reference (xref) is a drawing file linked (or attached) to another drawing. You cannot explode xrefs and their dependent blocks.

To explode an object

- 1 Click Modify menu ► Explode.
- 2 Select the objects to be exploded.
For most objects, exploding has no visible effect.

Modify toolbar



Command line: EXPLODE

Modify or Join Polylines

Additional editing operations are available for changing the shape of polyline objects. You can also join separate polylines.

You can edit polylines by closing and opening them and by moving, adding, or deleting individual vertices. You can straighten the polyline between any two vertices and toggle the linetype so that a dash appears before and after each vertex. You can set a uniform width for the entire polyline or control the width of each segment. You can also create a linear approximation of a spline from a polyline.

Joined Polyline Segments

You can join a line, an arc, or another polyline to an open polyline if their ends connect or are close to each other. If the ends are not coincident but are within a distance that you can set, called the *fuzz distance*, the ends are joined by either trimming them, extending them, or connecting them with a new segment.

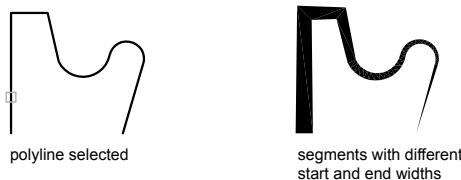
Properties of Modified Polylines

If the properties of several objects being joined into a polyline differ, the resulting polyline inherits the properties of the first object that you selected. If two lines meet a polyline in a Y shape, one of the lines is selected and joined to the polyline. Joining also causes an implicit decurve, with the program discarding the spline information of the original polyline and any polylines being joined to it. Once the joining is completed, you can fit a new spline to the resulting polyline.

Additional Editing Options for Polylines

In addition to the general editing operations available for most objects, there are additional options available for editing and joining polylines with PEDIT.

- **Close.** Creates the closing segment of the polyline, connecting the last segment with the first. The polyline is considered open unless you close it using the Close option.
- **Join.** Adds lines, arcs, or polylines to the end of an open polyline and removes the curve fitting from a curve-fit polyline. For an object to join the polyline, their endpoints must touch.
- **Width.** Specifies a new uniform width for the entire polyline. Use the Width option of the Edit Vertex option to change the starting and ending widths of segments.



- **Edit Vertex.** Marks the first vertex of the polyline by drawing an X on the screen. If you have specified a tangent direction for this vertex, an arrow is also drawn in that direction.
- **Fit.** Creates a smooth curve consisting of arcs joining each pair of vertices. The curve passes through all vertices of the polyline and uses any tangent direction you specify.
- **Spline.** Uses the vertices of the selected polyline as the control points, or frame, of a spline-fit polyline. The curve passes through the first and last control points unless the original polyline was closed.



- **Decurve.** Removes extra vertices inserted by an arc-fit or spline-fit polyline and straightens all segments of the polyline.
- **Ltype Gen.** Generates the linetype in a continuous pattern through the vertices of the polyline. When this option is turned off, the linetype is generated starting and ending with a dash at each vertex.

See also:

- “Trim or Extend Objects” on page 632
- “Break and Join Objects” on page 652

To modify a polyline

1 Click Modify menu ▶ Object ▶ Polyline.

2 Select the polyline to modify.

If the selected object is a line or an arc, the following prompt is displayed:

Object selected is not a polyline.

Do you want it to turn into one? <Y>: Enter **y** or **n**, or press ENTER

If you enter **y**, the object is converted into a single-segment 2D polyline that you can edit. You can use this operation to join lines and arcs into a polyline. When the PEDITACCEPT system variable is set to 1, this prompt is suppressed, and the selected object is automatically converted to a polyline.

3 Edit the polyline by entering one or more of the following options:

- Enter **c** (Close) to create a closed polyline.
- Enter **j** (Join) to join contiguous lines, arcs, or polylines.
- Enter **w** (Width) to specify a new uniform width for the entire polyline.
- Enter **e** (Edit Vertex) to edit a vertex.
- Enter **f** (Fit) to create a series of arcs joining each pair of vertices.
- Enter **s** (Spline) to create an approximation of a spline.

- Enter **d** (Decurve) to remove extra vertices inserted by a fit or spline curve and to straighten all segments of the polyline.
 - Enter **L** (Ltype Gen) to generate the linetype in a continuous pattern through the vertices of the polyline.
 - Enter **u** (Undo) to reverse actions back to the start of PEDIT.
- 4** Enter **x** (Exit) to end a command option. Press ENTER to exit the PEDIT command.

Modify II toolbar



Command line: PEDIT

To join polylines, lines, and arcs into a single polyline

- 1 Click Modify menu ▶ Object ▶ Polyline.
- 2 Select a polyline, line, or arc to edit. If you selected a line or arc, press ENTER to convert the selected line or arc into a polyline.
- 3 Enter **j** (Join).
- 4 Select one or more polylines, lines, or arcs that are located end to end.
Each selected polyline, line, or arc is now joined into a single polyline.
- 5 Press ENTER to end the command.

Modify II toolbar



Command line: PEDIT

To delete a vertex in a polyline

- 1 Click Modify menu ▶ Object ▶ Polyline.
- 2 Select a polyline.
- 3 Enter **e** (Edit vertex).
The first vertex is marked with an X. Use the Next option to move the X to the vertex preceding the one that you want to delete.
- 4 Enter **s** (Straighten).

5 Use the Next option to move the X to the vertex immediately following the one that you want to delete.

6 Enter **g** (Go).

The vertex on the polyline is deleted. The vertices on either side of the deleted vertex are joined by a straight polyline segment.

7 Enter **x** (Exit) to end editing vertexes.

8 Press ENTER to end the command.

Modify II toolbar



Command line: PEDIT

To taper the width of individual polyline segments

1 Click Modify menu ► Object ► Polyline.

2 Select the polyline to edit.

3 Enter **e** (Edit Vertex).

The first vertex is marked with an X. Move to the appropriate vertex with Next or Previous.

4 Enter **w** (Width).

5 Enter new starting and ending widths, and press ENTER to move to the next vertex. Repeat steps 4 and 5 for each segment.

6 Enter **u** (Undo) to reverse actions back to the start of PEDIT.

7 Enter **x** (Exit) to end editing vertexes.

8 Press ENTER to end the command.

Modify II toolbar



Command line: PEDIT

Modify Splines

Additional editing options are available for changing the shape of spline objects.

In addition to the general editing operations available for most objects, there are additional options available for editing splines with SPLINEDIT.

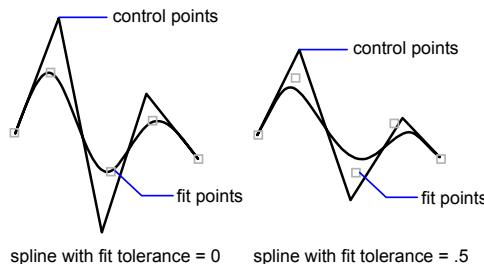
- **Fit Data.** Edits the fit point data that defines the spline, including changing the tolerance.
- **Close.** Changes an open spline into a continuous, closed loop.
- **Move Vertex.** Moves a fit point to a new location.
- **Refine.** Modifies a spline definition by adding and weighting control points and elevating the order of the spline.
- **Reverse.** Changes the direction of the spline.

You also can change the tolerance of the spline. Tolerance refers to how closely the spline fits the set of fit points you specify. The lower the tolerance, the more closely the spline fits the points.

Edit Splines with Grips

When you select a spline, grips are displayed on its fit points (the GRIPS system variable must be set to 1). You can use grips to modify the shape and location of the spline.

After certain operations, fit points are discarded and grips are displayed on control points instead. These operations include trimming the spline, moving the control points, and purging the fit data. If the spline's control frame is turned on (the SPLFRAME system variable is set to 1), grips are displayed on both the spline's control points and its fit points, when available.



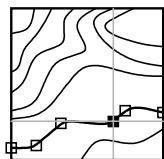
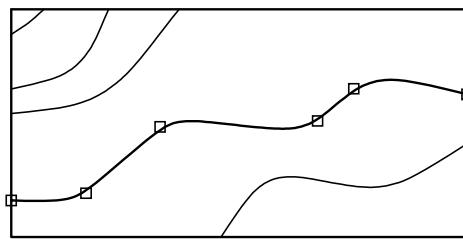
You can delete fit points of a spline, add fit points for greater accuracy, or move fit points to alter the shape of a spline. You can open or close a spline and edit the spline start and end tangents. Spline direction is reversible. You can change the *tolerance* of the spline also. Tolerance refers to how closely the

spline fits the set of fit points you specify. The lower the tolerance, the more closely the spline fits the points.

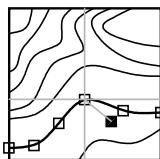
Refine the Shape of a Spline

You can refine a spline by increasing the number of control points in one portion of the spline or by changing the weight of specific control points. Increasing the weight of a control point pulls the spline more towards that point. You can also refine a spline by changing its order. A spline's order is the degree of the spline polynomial + 1. A cubic spline, for example, has order 4. The higher a spline's order, the more control points it has.

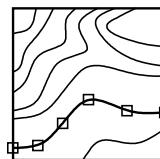
Consider the following example. You have created a spline to represent a geographic contour. Grips are turned on, and you need to move the fourth fit point to increase accuracy. When you select the spline, grips appear at the control points. If you created the spline by fitting it through a set of points, and you haven't purged this information using the Purge option of the SPLINEDIT command, and you select the Fit Data option, grips appear at the fit points on the selected spline instead of at the control points.



fourth fit point selected



fit point moved



result

See also:

["Break and Join Objects" on page 652](#)

To edit a spline

- 1 Click Modify menu ▶ Object ▶ Spline.
- 2 Select the spline to modify.
- 3 Edit the spline by entering one or more of the following options:
 - Enter **f** (Fit) to edit the fit data that defines the spline.
 - Enter **c** (Close) to change an open spline into a continuous, closed loop.
 - Enter **m** (Move Vertex) to move a fit point to a new location.
 - Enter **r** (Refine) to modify a spline definition by adding and weighting control points and elevating the order of the spline.
 - Enter **e** (Reverse) to reverse the direction of the spline.
 - Enter **u** (Undo) to cancel the last editing action.
- 4 Enter **x** (Exit) to end the command.

Modify II toolbar



Command line: SPLINEDIT

Modify Helices

You can use grips or the Properties palette to modify the shape and size of a helix.

You can use the grips on a helix to change the following:

- Start point
- Base radius
- Top radius
- Height
- Location

When you use a grip to change the base radius of a helix, the top radius scales to maintain the current ratio. Use the Properties palette to change the base radius independent of the top radius.

You can use the Properties palette to change other helix properties, such as

- Number of turns (Turns)
- Turn height
- Direction of the twist — clockwise (CW) or counterclockwise (CCW)

With the Constrain property, you can specify that the Height, Turns, or Turn Height properties of the helix are constrained. The Constrain property affects how the helix changes when the Height, Turns, or Turn Height properties are changed either through the Properties palette or through grip editing. The table below shows the behavior of the helix depending on which property is constrained.

Constrained property	Property to change	Effect on these helix properties		
		Height	Turns	Turn Height
Height	Height	Changed	Fixed	Changed
	Turns	Fixed	Changed	Changed
	Turn Height	Fixed	Changed	Changed
Turns	Height	Changed	Fixed	Changed
	Turns	Fixed	Changed	Changed
	Turn Height	Changed	Fixed	Changed
Turn Height	Height	Changed	Changed	Fixed
	Turns	Changed	Changed	Fixed
	Turn Height	Fixed	Changed	Changed

See also:

["Draw Helices" on page 447](#)

Modify Multilines

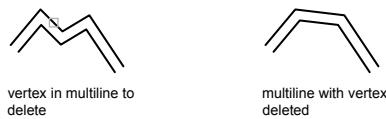
Multiline objects are composed of 1 to 16 parallel lines, called *elements*. To modify multilines or their elements, you can use common editing commands, a multiline editing command, and multiline styles.

Special multiline editing features are available with the MLEDIT command including the following:

- Add or delete a vertex
- Control the visibility of corner joints
- Control the style of intersection with other multilines
- Open or close gaps in a multiline object

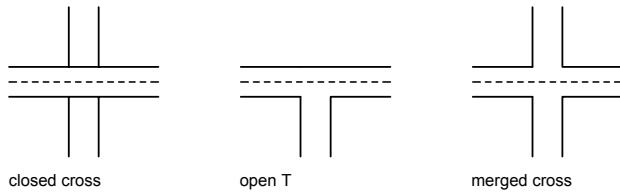
Add and Delete Multiline Vertices

You can add or delete any vertex in a multiline.



Edit Multiline Intersections

If you have two multilines in a drawing, you can control the way they intersect. Multilines can intersect in a cross or a T shape, and the crosses or T shapes can be closed, open, or merged.



Edit Multiline Styles

You can use MLSTYLE to edit multiline styles to change the properties of multiline elements or the end caps and background fill of subsequently created multilines.

Multiline styles control the number of line elements in a multiline and the color, linetype, linewidth, and offset of each element. You can also modify the display of joints, end caps, and background fill.

Multiline styles have the following limitations:

- You cannot edit the element and multiline properties of the STANDARD multiline style or any multiline style already used in the drawing.
- To edit an existing multiline style, you must do so *before* you draw any multilines in that style.

NOTE If you use MLSTYLE to create a multiline style without saving it, and then select another style or create a new style, the first MLSTYLE properties are lost. To maintain the properties, save each multiline style to an MLN file before creating a new one.

Use Common Editing Commands on Multilines

You can use most of the common editing commands on multilines *except*

- BREAK
- CHAMFER
- FILLET
- LENGTHEN
- OFFSET

To perform these operations, first use EXPLODE to replace the multiline object with separate line objects.

NOTE If you trim or extend a multiline object, only the first boundary object encountered determines the shape of the end of the multiline. A multiline cannot have a complex boundary at its endpoint.

See also:

["Draw Multiline Objects" on page 426](#)

To delete a vertex from a multiline

- 1 Click Modify menu ► Object ► Multiline.
- 2 In the Multiline Edit Tools dialog box, select Delete Vertex.
- 3 In the drawing, specify the vertex to delete. Press ENTER.

Command line: MLEDIT

To create a closed cross intersection

- 1 Click Modify menu ► Object ► Multiline.
- 2 In the Multiline Edit Tools dialog box, select Closed Cross.
- 3 Select the multiline for the foreground.
- 4 Select the multiline for the background.

The intersection is modified. You can continue selecting intersecting multilines to modify, or press ENTER to end the command. Press ENTER again to redisplay the Multiline Edit Tools dialog box.

To edit a multiline style

- 1 Click Format menu ► Multiline Style.
- 2 In the Multiline Styles dialog box, select the style name from the list. Click Modify.
- 3 Click Element Properties.
- 4 In the Modify Multiline Styles dialog box, change the settings as needed.
- 5 Click OK.
- 6 In the Multiline Styles dialog box, click Save to save the changes to the style in the MLN file.
- 7 Click OK.

Command line: MLSTYLE

Part 6

Work with 3D Models

Chapter 22 Create 3D Models

Chapter 23 Modify 3D Solids and Surfaces

Chapter 24 Create Sections and 2D Drawings from 3D Models

Create 3D Models

22

In this chapter

- Overview of 3D Modeling
- Create 3D Solids and Surfaces
- Check for Interferences within a Solid Model
- Create Meshes
- Create Wireframe Models
- Add 3D Thickness to Objects

Overview of 3D Modeling

With 3D modeling, you can create solid, wireframe, and mesh models of your design. 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
- Add lighting
- Create realistic renderings
- Navigate through the model
- Use the model to create an animation
- Perform engineering analysis
- Extract manufacturing data

Solid Modeling

When you create solid models, you use solids and surfaces as the building blocks for your model.

A solid object represents the entire volume of an object. Solids are the most informationally complete and least ambiguous of the 3D modeling types. Complex solid shapes are also easier to construct and edit than wireframes and meshes.

See also:

- [Enter 3D Coordinates](#)
- [Specify Workplanes in 3D \(UCS\)](#)
- [Use the Dynamic UCS with Solid Models](#)

No procedures for this topic.

Create 3D Solids and Surfaces

Overview of Creating 3D Solids and Surfaces

You can create basic 3D shapes (solid primitives): boxes, cones, cylinders, spheres, wedges, pyramids, and tori (donuts). You can then combine these shapes to create more complex solids by joining or subtracting them or finding their intersecting (overlapping) volume.

You can also create 3D solids and surfaces from existing objects through any of the following methods:

- Extruding objects
- Sweeping objects along a path
- Revolving objects around an axis
- Lofting through a set of curves
- Slicing a solid
- Converting planar objects with thickness into solids and surfaces

Solids and surfaces are displayed in the visual style that is applied to the viewport.

You can analyze solids for their mass properties (volume, moments of inertia, center of gravity, and so on). You can export data about a solid object to applications such as NC (numerical control) milling or FEM (finite element method) analysis. By exploding a solid, you can break it down to regions, bodies, surfaces, and wireframe objects.

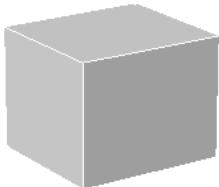
The ISOLINES system variable controls the number of tessellation lines used to visualize curved portions of the wireframe. The FACETRES system variable adjusts the smoothness of shaded and hidden-line objects.

By default, 3D solids record a *history* of their original forms. This history allows you to see the original forms that make up composite solids. For more information about History settings for solids and surfaces, see Display Original Forms of Composite Solids.

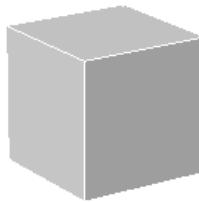
No procedures for this topic.

Create 3D Solid Primitives

Create a Solid Box



You can use the Cube option of the BOX command to create a box with sides of equal length.



If you use the Cube or Length option when you create a box, you can also specify the rotation of the box in the XY plane when clicking to specify the length.

You can also use the Center Point option to create a box using a specified center point.

To create a solid box

- 1 Click Draw menu » Modeling » Box.
- 2 Specify the first corner of the base.
- 3 Specify the opposite corner of the base.
- 4 Specify the height.

Modeling toolbar

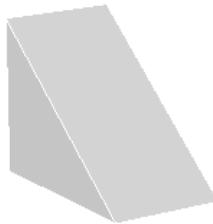


Command line: BOX

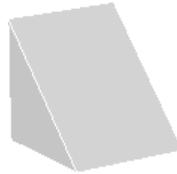
Create a Solid Wedge

You can create a solid wedge.

The base of the wedge is drawn parallel to the XY plane of the current UCS with the sloped face opposite the first corner. The height of the wedge is parallel to the Z axis.



You can use the Cube option of the WEDGE command to create a wedge with sides of equal length.



If you use the Cube or Length option when you create a wedge, you can also specify the rotation of the wedge in the XY plane when clicking to specify the length.

You can also use the Center Point option to create a wedge using a specified center point.

To create a solid wedge

- 1 Click Draw menu » Modeling » Wedge.
- 2 Specify the first corner of the base.
- 3 Specify the opposite corner of the base.
- 4 Specify the height of the wedge.

Modeling toolbar



Command line: WEDGE

3D Make panel, Wedge

Create a Solid Cone

By default, the cone's base lies on the XY plane of the current UCS. The height of the cone is parallel to the Z axis.

You can use the Axis Endpoint option of the CONE command to determine the height and orientation of the cone. The axis endpoint is the top point of the cone or the center of the top face if the Top Radius options is used. The axis endpoint can be located anywhere in 3D space

With the 3P (Three Points) option of the CONE command, you can define the base of the cone by specifying three points anywhere in 3D space.

Use the Top Radius option of the CONE command to create a cone frustum, which tapers to an elliptical or planar face.



To create a conical solid that requires a specific angle to define its sides, draw a 2D circle and 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.

To create a solid cone with a circular base

- 1 Click Draw menu » Modeling » Cone.
- 2 Specify the center point of the base.
- 3 Specify the radius or diameter of the base.
- 4 Specify the height of the cone.

Modeling toolbar



Command line: CONE

3D Make panel, Cone

To create a solid cone with an elliptical base

- 1 Click Draw menu » Modeling » Cone.
- 2 Enter **e** (Elliptical).
- 3 Specify an endpoint for first axis. This is the start point of the first axis.
- 4 Specify the other endpoint of the first axis. This is the endpoint of the first axis.
- 5 Specify the endpoint (length and rotation) of the second axis.
- 6 Specify the height of the cone.

Modeling toolbar



Command line: CONE

3D Make panel, Cone

To create a solid cone frustum

- 1 Click Draw menu » Modeling » Cone.
- 2 Specify the center point of the base.
- 3 Specify the radius or diameter of the base.
- 4 Enter **t** (Top radius).
- 5 Specify the top radius.
- 6 Specify the height of the cone.

Modeling toolbar



Command line: CONE

3D Make panel, Cone

To create a solid cone with the height and orientation specified by the axis endpoint

- 1 Click Draw menu » Modeling » Cone.
- 2 Specify the center point of the base.
- 3 Specify the radius or diameter of the base.
- 4 On the command line, enter **a**.
- 5 Specify the axis endpoint of the cone.

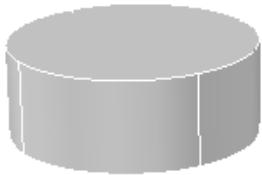
Modeling toolbar



Command line: CONE

3D Make panel, Cone

Create a Solid Cylinder



You can use the Axis Endpoint option of the CYLINDER command to determine the height and orientation of the cone. The axis endpoint is the center point of the top plane of the cylinder. The axis endpoint can be located anywhere in 3D space.

With the 3P (Three Points) option of the CYLINDER command, you can define the base of the cylinder by specifying three points anywhere in 3D space.

If you want to construct a cylindrical form with special detail, such as grooves along its sides, create a 2D profile of its base with a closed PLINE and use EXTRUDE to define its height along the Z axis. This method, however, creates an extruded solid, not a true solid cylinder primitive.

To create a solid cylinder with a circular base

- 1 Click Draw menu » Modeling » Cylinder.
- 2 Specify the center point of the base.

- 3** Specify the radius or diameter of the base.
- 4** Specify the height of the cylinder.

Modeling toolbar



Command line: CYLINDER

3D Make panel, Cylinder

To create a solid cylinder with an elliptical base

- 1** Click Draw menu » Modeling » Cylinder.
- 2** Enter **e** (Elliptical).
- 3** Specify an endpoint for first axis. This is the start point of the first axis.
- 4** Specify the other endpoint of the first axis. This is the endpoint of the first axis.
- 5** Specify the endpoint (length and rotation) of the second axis.
- 6** Specify the height of the cylinder.

Modeling toolbar



Command line: CYLINDER

3D Make panel, Cylinder

To create a solid cylinder with the height and orientation specified by the axis endpoint

- 1** Click Draw menu » Modeling » Cylinder.
- 2** Specify the center point of the base.
- 3** Specify the radius or diameter of the base.
- 4** On the command line, enter **a**.
- 5** Specify the axis endpoint of the cylinder.

Modeling toolbar



Command line: CYLINDER

3D Make panel, Cylinder

Create a Solid Sphere

When you specify the center point, the sphere is positioned so that its central axis is parallel to the Z axis of the current user coordinate system (UCS).



You can also use any of the following options in the SPHERE command to define the sphere:

- **3P (Three Points).** Defines the circumference of the sphere by specifying three points anywhere in 3D space. The three specified points also define the plane of the circumference.
- **2P (Two Points).** Defines the circumference of the sphere by specifying two points anywhere in 3D space. The plane of the circumference is defined by the Z value of the first point.
- **TTR (Tangent, Tangent, Radius).** Defines the sphere with a specified radius tangent to two objects. The specified tangency points are projected onto the current UCS.

To create a solid sphere

- 1 Click Draw menu » Modeling » Sphere.
- 2 Specify the center of the sphere.
- 3 Specify the radius or diameter of the sphere.

Modeling toolbar



Command line: SPHERE

3D Make panel, Sphere

To create a solid sphere defined by three points

- 1 Click Draw menu » Modeling » Sphere.
- 2 Enter **3p**.
- 3 Specify the first point.
- 4 Specify the second point.
- 5 Specify the third point.

Modeling toolbar



Command line: SPHERE

3D Make panel, Sphere

Create a Solid Pyramid

You can create a solid pyramid. You can define the number of sides for a pyramid from 3 to 32.



You can use the Axis Endpoint option of the PYRAMID command to specify the endpoint location of the pyramid axis. This endpoint is the top point of the pyramid or the center of the top face if the Top Radius option is used. The axis endpoint can be located anywhere in 3D space. The axis endpoint defines the length and orientation of the pyramid.

You can use the Top Radius option to create a pyramid frustum, which tapers to a planar face that has the same number of sides as the base.



To create a solid pyramid

- 1 Click Draw menu » Modeling » Pyramid.
- 2 Specify the center point of the base.
- 3 Specify the radius or diameter of the base.
- 4 Specify the height of the pyramid.

Modeling toolbar



Command line: PYRAMID

3D Make panel, Pyramid

To create a solid pyramid frustum

- 1 Click Draw menu » Modeling » Pyramid.
- 2 Specify the center point of the base.
- 3 Specify the radius or diameter of the base.
- 4 Enter **t** (Top radius).
- 5 Specify the top radius.
- 6 Specify the height of the pyramid.

Modeling toolbar



Command line: PYRAMID

3D Make panel, Pyramid

Create a Solid Torus



A torus is defined by two radius values, one for the tube and the other for the distance from the center of the torus to the center of the tube.

With the 3P (Three Points) option of the TORUS command, you can define the circumference of the torus by specifying three points anywhere in 3D space.

The torus is drawn parallel to and bisected by the XY plane of the current UCS (this may not be true if you use the 3P [Three Points] option of the TORUS command).

A torus may be self-intersecting. A self-intersecting torus has no center hole because the radius of the tube is greater than the radius of the torus.

To create a solid torus

- 1 Click Draw menu » Modeling » Torus.
- 2 Specify the center of the torus.
- 3 Specify the radius or diameter of the torus.
- 4 Specify the radius or diameter of the tube.

Modeling toolbar

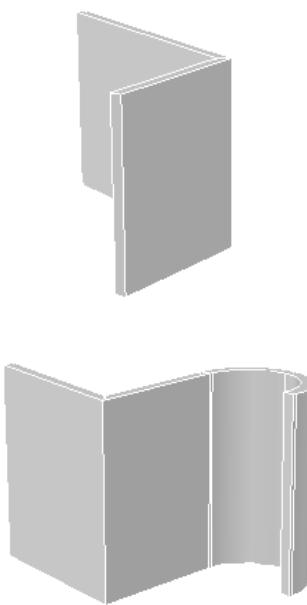


Command line: TORUS

3D Make panel, Torus

Create a Polysolid

A polysolid is drawn the same way you draw a polyline. By default, a polysolid always has a rectangular profile. You can specify the height and width of the profile. Use POLYSOLID to create walls in your model.



When drawing a polysolid, you can use the Arc option to add arc segments to the polysolid. You can use the Close option to close the solid between the first and last specified points.

The PSOLIDWIDTH system variable sets the default width for polysolids. The PSOLIDHEIGHT system variable sets the default height.

When you use an existing object to create the polysolid, the DELOBJ system variable controls whether the path is automatically deleted when the solid is created or whether you are prompted to delete the object.

Polysolids are swept solids (solids that are drawn with a specified profile along the specified path) and display as swept solids in the Properties palette.

To draw a polysolid

- 1 Click Draw menu » Modeling » Polysolid.
- 2 Specify a start point.
- 3 Specify the next point.
- 4 Repeat step 3 to complete the desired solid.
- 5 Press ENTER.

Modeling toolbar



Command line: POLYSOLID

3D Make panel, Polysolid

To create a polysolid from an existing object

- 1 Click Draw menu » Modeling » Polysolid.
- 2 Enter **o**, and then press ENTER.
- 3 Select a line, 2D polyline, arc, or circle.

After the solid is created, the original object may be deleted or retained, depending on the setting of the DELOBJ system variable.

Modeling toolbar

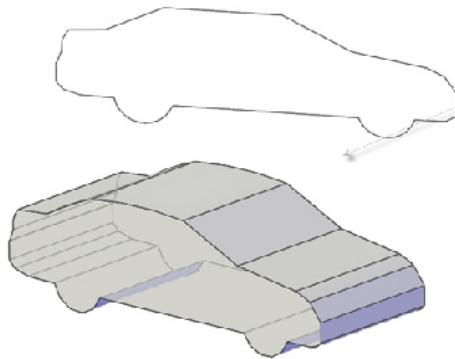


Command line: POLYSOLID

3D Make panel, Polysolid

Create Solids and Surfaces from Lines and Curves

Extrude Objects



If you extrude a closed object, the resulting object is a solid. If you extrude an open object, the resulting object is a surface.

You can extrude the following objects and subobjects:

- Lines
- Arcs
- Elliptical arcs
- 2D polylines
- 2D splines
- Circles
- Ellipses
- 3D faces
- 2D solids
- Traces
- Regions
- Planar surfaces

- Planar faces on solids

You cannot extrude the following:

- Polylines that have crossing or self-intersecting segments
- Objects contained within a block

If a selected polyline has width, the width is ignored and the polyline is extruded from the center of the polyline path. If a selected object has thickness, the thickness is ignored.

If you want to create a solid from a profile using lines or arcs, use the Join option of the PEDIT command to convert them to a single polyline object. You can also convert the objects to a region before you use EXTRUDE.

When you extrude objects, you can specify any of the following options:

- Path
- Taper angle
- Direction

Specify a Path for an Extruded Solid

With the Path option, you can specify an object as the path for the extrusion. The profile of the selected object is extruded along the chosen path to create a solid or surface. For best results, it is recommended that the path lie on or within the boundary of the object being extruded.

Extruding is different from sweeping. When you extrude a profile along a path, the path is moved to the profile if it does not already intersect the profile. Then the profile is swept along the path.

NOTE Using a path with the SWEEP command provides greater control and better results.

The extruded solid starts from the plane of the profile and ends on a plane perpendicular to the path at the path's endpoint.

The following objects can be paths:

- Lines
- Circles
- Arcs

- Ellipses
- Elliptical arcs
- 2D polylines
- 3D polylines
- 2D splines
- 3D splines
- Edges of solids
- Edges of surfaces
- Helixes

Specify a Taper Angle for an Extruded Solid

Tapering the extrusion is useful specifically for parts that need their sides defined along an angle, such as a mold used to create metal products in a foundry. Avoid using large tapered angles. If the angle is too large, the profile can taper to a point before it reaches the specified height.

Specify a Direction for an Extruded Solid

With the Direction option, you can specify the length and direction of the extrusion by specifying two points.

To extrude an object

- 1 Click Draw menu » Modeling » Extrude.
- 2 Select the object(s) to extrude, and then press ENTER.
- 3 Specify the height.

After the extrusion, the original object(s) may be deleted or retained, depending on the setting of the DELOBJ system variable.

Modeling toolbar



Command line: EXTRUDE

3D Make panel, Extrude

To extrude an object along a path

- 1 Click Draw menu » Modeling » Extrude.
- 2 Select the object(s) to extrude.
- 3 Enter **p** (Path), and then press ENTER.
- 4 Select the object to use as the path.

After the extrusion, the original object(s) may be deleted or retained, depending on the setting of the DELOBJ system variable.

Modeling toolbar



Command line: EXTRUDE

3D Make panel, Extrude

Create a Solid or Surface by Sweeping

With the SWEEP command, you can create a new solid or surface by sweeping an open or closed planar curve (profile) along an open or closed 2D or 3D path.

If you sweep a closed curve along a path the result is a solid.

If you sweep an open curve along a path, the result is a surface.

Sweeping is different from extruding. When you sweep a profile along a path, the profile is moved and aligned normal (perpendicular) to the path. Then the profile is swept along the path.

TIP To sweep a profile such as a closed polyline along a helix, move or rotate the profile into place and turn off the Alignment option in the SWEEP command. If you get a modeling error, make sure that the result will not intersect itself.

When you sweep objects, they can be twisted or scaled as they are swept. You can also use the Properties palette to specify the following properties for the profile after it's been swept:

- Profile Rotation
- Scale Along Path
- Twist Along Path
- Bank (natural rotation)

NOTE The Properties palette will not allow changes to be made to these properties if the Alignment option was turned off when the profile was swept, or if the change would result in a modeling error such as a self-intersecting solid. No warning message is displayed.

You can sweep more than one object at a time, but they must all lie on the same plane.

You can use the following objects and paths when creating a swept solid or surface:

Objects That Can Be Swept (Profiles)	Objects That Can Be Used as a Sweep Path
Line	Line
Arc	Arc
Elliptical arc	Elliptical arc
2D polyline	2D polyline
2D spline	2D spline
Circle	Circle
Ellipse	Ellipse
3D face	3D spline
2D solid	3D polyline
Trace	Helix
Region	Edges of solids or surface
Planar surface	
Planar faces of solid	

The DELOBJ system variable controls whether the profile and sweep path are automatically deleted when the solid or surface is created or whether you are prompted to delete the profile and path.

See also:

[Draw Helixes](#)

To create a solid or surface by sweeping an object along a path

- 1 Click Draw menu » Modeling » Sweep.

- 2** Select objects to sweep.
- 3** Press ENTER.
- 4** Select a sweep path.

After the sweep, the original objects are deleted or retained, depending on the setting of the DELOBJ system variable.

Modeling toolbar



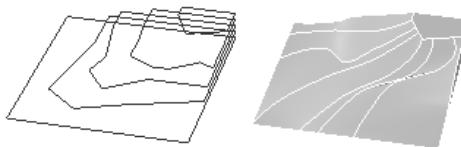
Command line: SWEEP

3D Make panel, Sweep

Create a Solid or Surface by Lofting

With the LOFT command, you can create a 3D solid or surface by lofting (drawing a solid or surface) through a set of two or more cross-section curves.

If you loft through a set of a closed cross-section curves, the result is a solid.



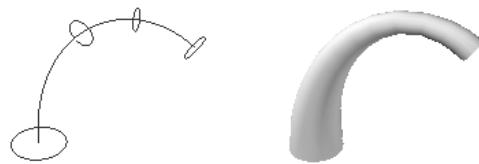
Cross sections

Lofted solid

If you loft through a set of open cross-section curves, the result is a surface.

NOTE The curves that you use when lofting must all be open or closed. You cannot use a selection set that includes both open and closed curves.

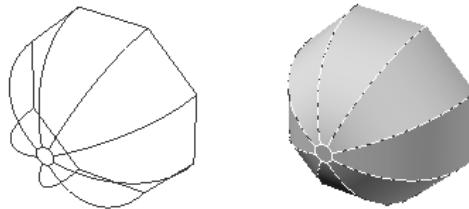
You can specify a path for the loft operation. Specifying a path gives you more control over the shape of the lofted solid or surface. It is recommended that the path curve start on the plane of the first cross section and end on the plane of the last cross section.



Cross sections
with path curve

Lofted solid

You can also specify guide curves when lofting. Guide curves are another way to control the shape of the lofted solid or surface. You can use guide curves to control how points are matched up on corresponding cross sections to prevent undesired results, such as wrinkles in the resulting solid or surface.



Cross sections
with guide curves

Lofted solid

Each guide curve must meet the following criteria:

- Intersects each cross section
- Starts on the first cross section
- Ends on the last cross section

You can select any number of guide curves for the lofted surface or solid.

When you use only cross sections to create a lofted surface or solid, you can also use the options in the Loft Settings dialog box to control the shape of the surface or solid.

You can use the following table shows objects you can use when creating a lofted solid or surface.

Objects That Can Be Used as Cross Sections	Objects That Can Be Used as a Loft Path	Objects That Can Be Used as Guides
Line	Line	Line
Arc	Arc	Arc
Elliptical arc	Elliptical arc	Elliptical arc
2D polyline	Spline	2D spline
2D spline	Helix	3D spline
Circle	Circle	2D polyline
Ellipse	Ellipse	3D polyline
Points (first and last cross section only)	2D polyline	
		3D polyline

The DELOBJ system variable controls whether the cross sections, paths, and guides are automatically deleted when the solid or surface is created or whether you are prompted to delete the profile(s) and path.

To create a solid or surface by lofting through a set of cross sections

- 1 Click Draw menu » Modeling » Loft.
- 2 Select cross sections in the order in which you want the solid or surface to pass through the cross sections.
- 3 Press ENTER.
- 4 Do one of the following:

- Press ENTER or enter **c** to use cross sections only.

The Loft Settings dialog box is displayed. Use the options in the dialog box to control the shape of the solid or surface. Change the desired settings, and click Preview to see a preview of the surface or solid.

Click OK when finished.

- Enter **g** to select guide curves. Select the guide curves, and then press ENTER.
- Enter **p** to select a path. Select a path, and then press ENTER.

After the loft, the original objects may be deleted or retained, depending on the setting of the DELOBJ system variable.

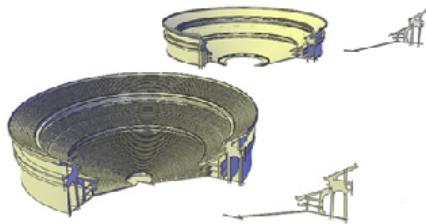
Modeling toolbar



Command line: LOFT

3D Make panel, Loft

Create a Solid or Surface by Revolving

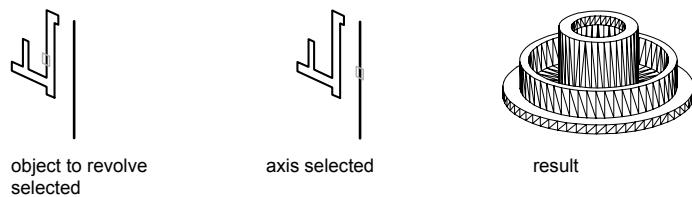
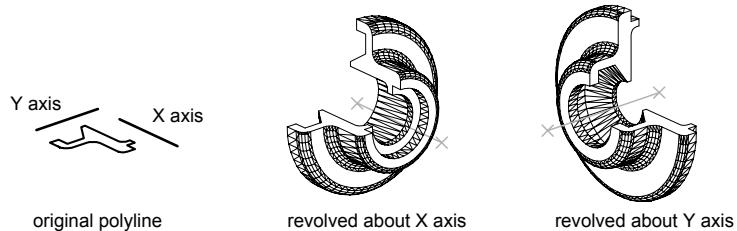


If you revolve a closed object, the result is a solid. If you revolve an open object, the result is a surface.

You can revolve more than one object at a time.

When you revolve objects, you can specify any of the following as the axis about which to revolve the objects:

- Axis defined by two points you specify
- X axis
- Y axis
- Z axis
- Axis defined by an object (Object option)



You can use the following objects when you use REVOLVE:

Objects that Can Be Revolved	Objects that Can Be Used as an Axis of Revolution
Line	Line
Arc	Linear polyline segment
Elliptical arc	Linear edge of a surface
2D polyline	Linear edge of a solid
2D spline	
Circle	
Ellipse	
3D faces	
2D solid	
Trace	
Region	
Planar surface	

Objects that Can Be Revolved	Objects that Can Be Used as an Axis of Revolution
Planar face of solid	

If you want to create a solid using a profile made up of lines or arcs that meet a polyline, use the PEDIT Join option to convert them to a single polyline object before you use REVOLVE. If you don't convert these objects to a single polyline, a surface will be created when you revolve them.

You cannot use REVOLVE on objects contained within a block or polylines that have crossing or intersecting segments.

To revolve objects about an axis

- 1 Click Draw menu » Modeling » Revolve.
- 2 Select the object(s) to revolve.
- 3 Specify the start point and endpoint of the axis of revolution.
Specify the points so that the object is on one side of the axis points you specify. The positive axis direction is from the start point to the endpoint.
- 4 Specify the angle of revolution.

Modeling toolbar



Command line: REVOLVE

3D Make panel, Revolve

Create Solids and Surfaces from Objects

You can create surfaces from existing objects in your drawing.

- 2D solids
- Regions
- Bodies
- Open, zero-width polylines with thickness
- Lines with thickness
- Arcs with thickness

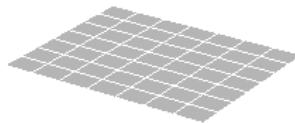
■ Planar 3D faces

You can create surfaces from 3D solids with curved faces, such as a cylinder, with the EXPLODE command.

You can use the PLANESURF command to create a planar surface. Use either of the following methods:

- Select one or more objects that form one or more enclosed areas
- Specify the opposite corners of a rectangle

When you specify the corners of the surface, the surface is created parallel to the workplane.



You can use the CONVTOSOLID command to convert the following objects into extruded 3D solids:

- Uniform-width wide polylines with thickness
- Closed, zero-width polylines with thickness
- Circles with thickness

NOTE You cannot use CONVTOSOLID with polylines that contain vertices with 0 width or that contain segments of variable width.

The DELOBJ system variable controls whether the objects you select are automatically deleted when the surface is created or whether you are prompted to delete the objects.

To convert one or more objects to surfaces

- 1 Click Modify menu » 3D Operations » Convert to Surface.
- 2 Select the objects you want to convert.
- 3 Press ENTER.

Command line: CONVTOSURFACE

3D Make panel (click icon to expand), Convert to Surface

To create a planar surface from an existing object

- 1 Click Draw menu » Modeling » Planar Surface.
- 2 Enter **o**.
- 3 Select an object.
- 4 Press ENTER.

Modeling toolbar



Command line: PLANESURF

3D Make panel, Planar Surface

To create a planar surface by specifying the corners of the surface

- 1 Click Draw menu » Modeling » Planar Surface.
- 2 Specify the first corner of the surface.
- 3 Specify the second corner of the surface.

Modeling toolbar



Command line: PLANESURF

3D Make panel, Planar Surface

To convert objects with thickness to extruded solids

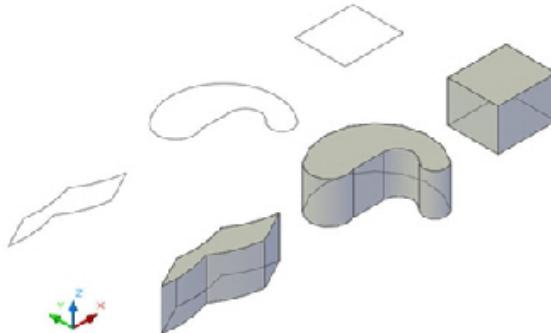
- 1 Click Modify menu » 3D Operations » Convert to Solid.
- 2 Select one or more of the following types of objects with thickness.
 - Uniform-width polylines
 - Closed, 0-width polylines
 - Circles
- 3 Press ENTER.

Command line: CONVTOSOLID

3D Make panel (click icon to expand), Convert to Solid

Create Solids from Surfaces

You can create a 3D solid from any surface type by thickening the surface.



The DELOBJ system variable controls whether the object you select is automatically deleted when the surface is created or whether you are prompted to delete the object.

To convert one or more surfaces to solids

- 1 Click Modify menu » 3D Operations » Thicken.
- 2 Select the surfaces you want to thicken.
- 3 Press ENTER.
- 4 Specify a thickness for the solid.
- 5 Press ENTER.

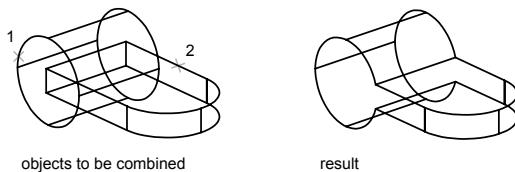
Command line: THICKEN

3D Make panel, Thicken Surface

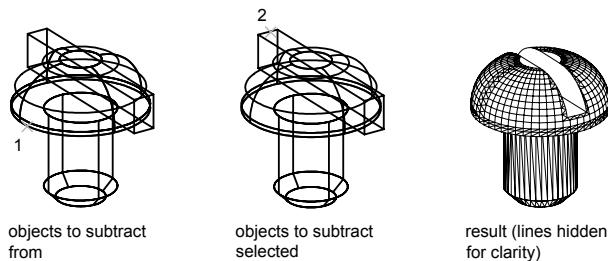
Create Composite Solids

By default, 3D solids record a *history* of their original forms. This history allows you to see the original forms that make up composite solids. For more information about History settings for solids and surfaces, see Display Original Forms of Composite Solids.

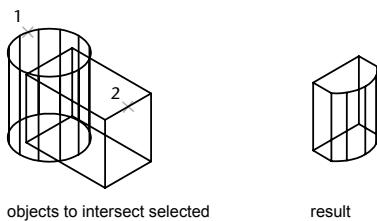
With UNION, you can combine the total volume of two or more solids or two or more regions into a composite object.



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.



With INTERSECT, you can create a composite solid from the common volume of two or more overlapping solids. INTERSECT removes the nonoverlapping portions and creates a composite solid from the common volume.



To combine solids

- 1 Click Modify menu » Solid Editing » Union.

- 2** Select the objects to combine.
- 3** Press ENTER.

Solid Editing toolbar



Command line: UNION

3D Make panel, Union

To subtract solids from one another

- 1** Click Modify menu » Solid Editing » Subtract.
- 2** Select the objects to subtract from.
- 3** Press ENTER.
- 4** Select the objects to subtract.
- 5** Press ENTER.

Solid Editing toolbar



Command line: SUBTRACT

3D Make panel, Subtract

To create a solid from the intersection of two or more other solids

- 1** Click Modify menu » Solid Editing » Intersect.
- 2** Select the objects to intersect.
- 3** Press ENTER.

Solid Editing toolbar

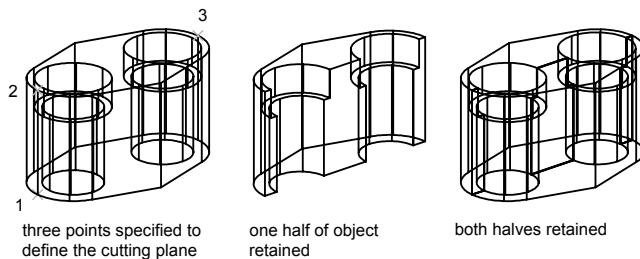


Command line: INTERSECT

3D Make panel, Intersect

Create Solids by Slicing

You can create new solids by slicing existing ones. You can define the cutting plane a number of ways, including specifying points or selecting a surface or planar object.



The following objects can be used as the cutting plane:

- Surfaces
- Circles
- Ellipses
- Circular or elliptical arcs
- 2D splines
- 2D polyline segments

See also:

[Create 3D Models](#)

To slice solids

- 1 Click Modify menu » 3D Operations » Slice.
- 2 Select the objects to slice.
- 3 Press ENTER.
- 4 Specify two points to define the cutting plane.
- 5 Specify which side to retain, or enter **b** to retain both sides.

Command line: SLICE

3D Make panel (click icon to expand), Slice

To slice solids with a planar object

- 1 Click Draw menu » 3D Operations » Slice.
- 2 Select the objects to slice.
- 3 Press ENTER.
- 4 On the command line, enter **o**.
- 5 Press ENTER.
- 6 Select a circle, ellipse, arc, 2D spline, or 2D polyline to use as the cutting plane.
- 7 Specify which side to retain, or enter **b** to retain both sides.

Command line: SLICE

3D Make panel (click icon to expand), Slice

To slice solids with a surface

- 1 Click Draw menu » 3D Operations » Slice.
- 2 Select the objects to slice.
- 3 Press ENTER.
- 4 On the command line, enter **s**.
- 5 Press ENTER.
- 6 Select a surface to use as the cutting plane.
- 7 Specify which side to retain, or enter **b** to retain both sides.

Command line: SLICE

3D Make panel (click icon to expand), Slice

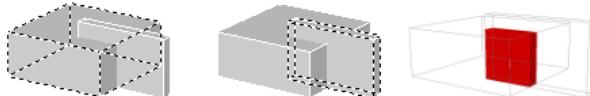
Check for Interferences within a Solid Model

You can use the INTERFERE command to check for interferences (areas where 3D solids intersect or overlap) within a solid model by comparing two sets of objects or by checking all the solids against one another.

You can also use the INTERFERE command with blocks that contain 3D solids and with nested solids within blocks.

When you use the INTERFERE command, temporary solids are created and highlighted where solids intersect.

If you define a single selection set (one set of objects), INTERFERE checks all the solids in the set against one another. If you define two selection sets (two sets of objects), INTERFERE checks the solids in the first selection set against those in the second selection set.



Once you start checking for interferences, you can use the Interference Checking Dialog Box to cycle through and zoom to interference objects. You can also specify that the temporary interference objects that are created during interference checking are deleted when you close the dialog box.

You can specify the display of interference objects by using the options in the Interference Settings Dialog Box.

To check for interferences within a solid model

- 1 Click Modify menu » 3D Operations » Interference Checking.
- 2 Select the first set of objects in the model.
- 3 Press ENTER.
- 4 Select the second set of objects in the model.
- 5 Press ENTER.
- 6 (Optional) In the Interference Checking dialog box, click Next and Previous to cycle through the interference objects.
- 7 (Optional) If you don't want interference objects to be deleted when you close the Interference Checking dialog box, clear Delete Interference Objects Created on Close.
- 8 Click Close.

Interference objects are deleted if Delete Interference Objects on Close is selected.

Command line: INTERFERE

3D Make panel (click icon to expand), Interference Checking

To change the display of interference objects

- 1 Click Modify menu » 3D Operations » Interference Checking.
- 2 Enter **s**.
- 3 Press ENTER.
- 4 In the Interference Settings dialog box, change any of the settings.
- 5 Click OK.

Command line: INTERFERE

3D Make panel (click icon to expand), Interference Checking

Create Meshes

You can create polygonal mesh forms. Because the faces of the mesh are planar, the mesh can only approximate curved surfaces.

You can use meshes if you need hiding, shading, and rendering capabilities that wireframe models cannot provide but do not need the physical properties that solid models provide (mass, volume, center of gravity, moments of inertia, and so on).

You can also use meshes to create geometry with unusual mesh patterns, such as a 3D topographical model of mountainous terrain.

The visual style controls how a mesh is displayed, either wireframe or shaded. (VISUALSTYLES command)

You can create several types of meshes.

- **3D face.** 3DFACE creates a planar mesh with either three or four sides.
- **Ruled mesh.** RULESURF creates a polygon mesh representing the ruled surface between two lines or curves.
- **Tabulated mesh.** TABSURF creates a polygon mesh representing a general tabulated surface defined by the extrusion of a line or curve (called a path curve) in a specified direction and distance (called a direction vector).
- **Revolved mesh.** REVSURF creates a polygon mesh approximating a surface of revolution by rotating a path curve or profile (lines, circles, arcs, ellipses, elliptical arcs, polylines, or splines, closed polylines, polygons, closed splines, or donuts) about a specified axis.
- **Edge-defined mesh.** EDGESURF creates a polygon mesh approximating a Coons surface patch mesh from four adjoining edges. A Coons surface

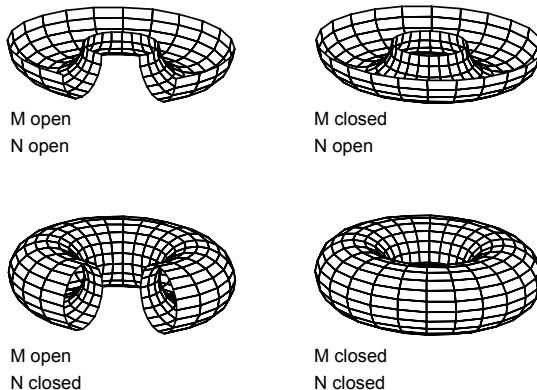
patch mesh is a bicubic surface interpolated between four adjoining edges (which can be general space curves).

- **Predefined 3D mesh.** 3D creates three-dimensional polygon mesh objects in common geometric shapes, including boxes, cones, spheres, tori, wedges, and pyramids.
- **General meshes.** 3DMESH and PFACE create three-dimensional polygon mesh objects in any shape.

Understand Mesh Construction

The mesh density controls the number of facets, and is defined in terms of a matrix of M and N vertices, similar to a grid consisting of columns and rows. M and N specify the column and row position, respectively, of any given vertex.

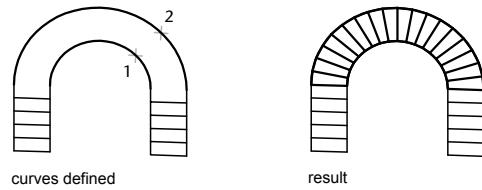
A mesh can be open or closed. A mesh is open in a given direction if the start and end edges of the mesh do not touch, as shown in the following illustrations.



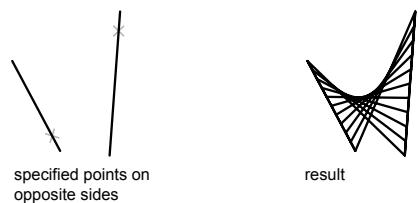
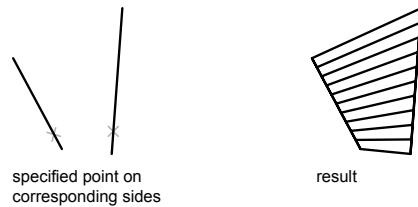
There are several methods for creating meshes.

Create a Ruled Mesh

With RULESURF, you can create a mesh between two lines or curves. You can 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. Pairs of objects to be used as the “rails” of a ruled mesh must both be either open or closed. You can pair a point object with either an open or a closed object.

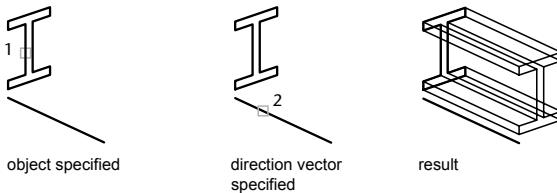


You can specify any two points on closed curves to complete RULESURF. For open curves, construction of the ruled mesh is based on the locations of the specified points on the curves.



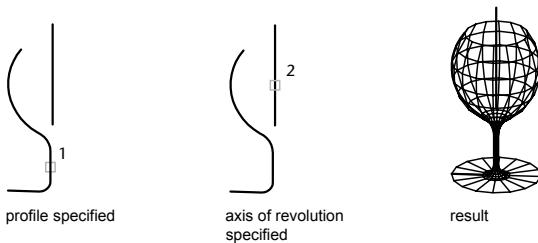
Create a Tabulated Mesh

With the TABSURF command, you can create a mesh representing a general tabulated surface defined by a path curve and a direction vector. The path curve can be a line, arc, circle, ellipse, elliptical arc, 2D polyline, 3D polyline, or spline. The direction vector can be a line or an open 2D or 3D polyline. TABSURF creates the mesh as a series of parallel polygons running along a specified path. You must have the original object and the direction vector already drawn, as shown in the following illustrations.



Create a Revolved Mesh

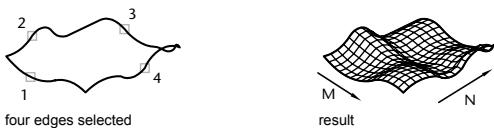
Use the REVSURF command to create a revolved mesh by rotating a profile of the object about an axis. REVSURF is useful for mesh forms with rotational symmetry.



The profile is called a path curve, which can be any combination of lines, circles, arcs, ellipses, elliptical arcs, polylines, or splines, closed polylines, polygons, closed splines, or donuts.

Create an Edge-Defined 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, and elliptical arcs, and they must form a closed loop and share endpoints. A Coons patch is a bicubic surface (one curve in the *M* direction and another in the *N* direction) interpolated between the four edges.

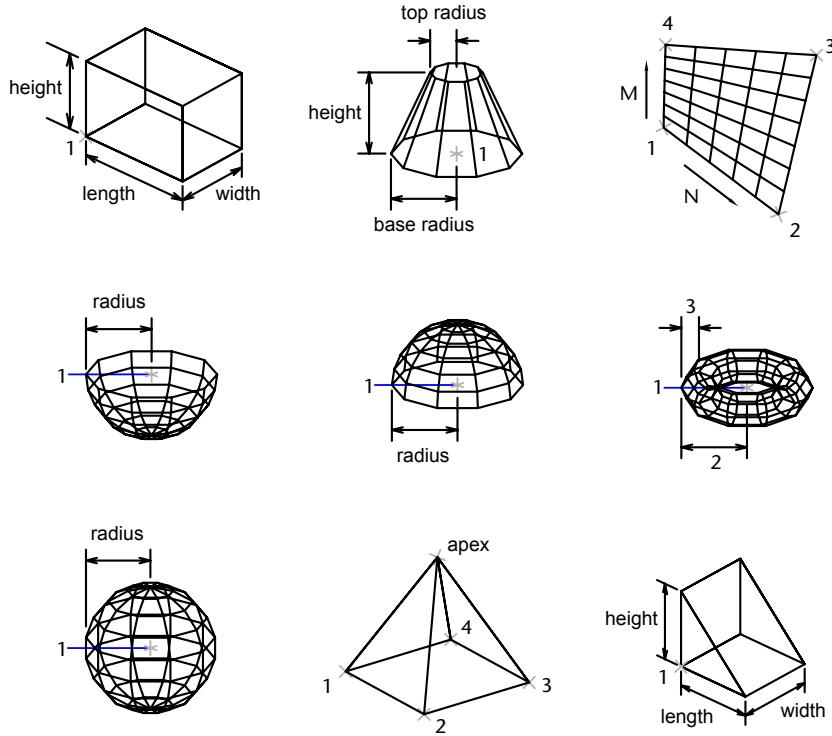


Create a Predefined 3D Mesh

The 3D command creates the following 3D shapes: boxes, cones, dishes, domes, meshes, pyramids, spheres, tori (donuts), and wedges.

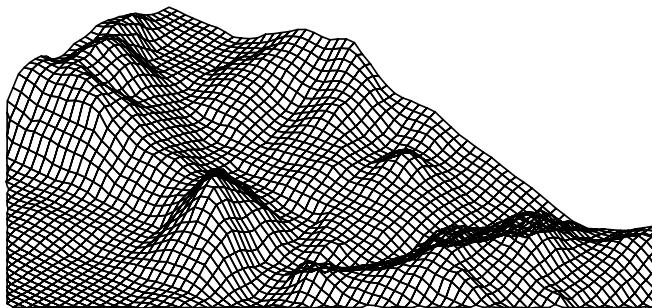
To view the objects you are creating with the 3D command more clearly, set a viewing direction with 3DORBIT, DVIEW, or VPOINT.

In the following illustrations, the numbers indicate points you specify to create the mesh.



Create a Rectangular Mesh

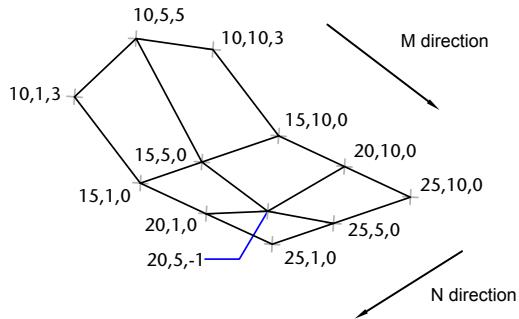
With the 3DMESH command, you can create polygon meshes that are open in both the *M* and *N* directions (similar to the *X* and *Y* axes of an *XY* plane). You can close the meshes with PEDIT. You can use 3DMESH to construct very irregular meshes. In most cases, you can use 3DMESH in conjunction with scripts or AutoLISP routines when you know the mesh points.



In the following example of text on the command line, you enter the coordinate values for each vertex to create the mesh in the illustration.

Example:

```
Command: 3dmesh
Mesh M size: 4
Mesh N size: 3
Vertex (0, 0): 10,1, 3
Vertex (0, 1): 10, 5, 5
Vertex (0, 2): 10,10, 3
Vertex (1, 0): 15,1, 0
Vertex (1, 1): 15, 5, 0
Vertex (1, 2): 15,10, 0
Vertex (2, 0): 20,1, 0
Vertex (2, 1): 20, 5, -1
Vertex (2, 2): 20,10, 0
Vertex (3, 0): 25,1, 0
Vertex (3, 1): 25, 5, 0
Vertex (3, 2): 25,10, 0
```



Create a Polyface Mesh

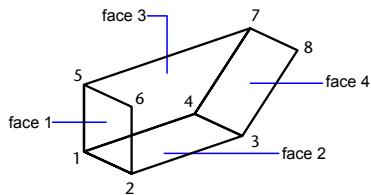
The PFACE command produces a polyface (polygon) mesh, with each face capable of having numerous vertices. PFACE is typically used by applications rather than by direct user input.

Creating a polyface mesh is similar to creating a rectangular mesh. To create a polyface mesh, you specify coordinates for its vertices. You then define each face by entering vertex numbers for all the vertices of that face. As you create the polyface mesh, you can set specific edges to be invisible, assign them to layers, or give them colors.

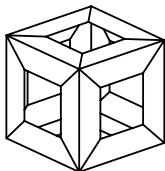
To make the edge invisible, enter the vertex number as a negative value. For instance, to make the edge between vertices 5 and 7 invisible in the following illustration, you enter the following:

Face 3, vertex 3: -7

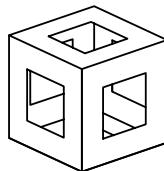
In the illustration, face 1 is defined by vertices 1, 5, 6, and 2. Face 2 is defined by vertices 1, 4, 3, and 2. Face 3 is defined by vertices 1, 4, 7, and 5, and face 4 is defined by vertices 3, 4, 7, and 8.



You can control the display of invisible edges with the SPLFRAME system variable. If SPLFRAME is set to a nonzero value, the invisible edges become visible and can then be edited. If SPLFRAME is set to 0, the invisible edges remain invisible.



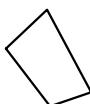
SPLFRAME = 1



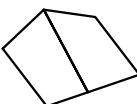
SPLFRAME = 0

To create a rectangular mesh

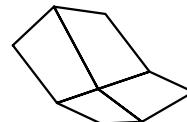
- 1 Click Draw menu ▶ Modeling ▶ Meshes ▶ 3D Mesh.
- 2 Specify the *M* size, using an integer from 2 through 256.
- 3 Specify the *N* size, using an integer from 2 through 256.
- 4 Specify the vertex points as prompted. Specifying the last vertex point completes the mesh.



Mesh M size: 2
Mesh N size: 2



Mesh M size: 2
Mesh N size: 3



Mesh M size: 3
Mesh N size: 3

Command line: 3DMESH

To create a ruled mesh

- 1 Click Draw menu ▶ Modeling ▶ Meshes ▶ Ruled Mesh.
- 2 Select the first defining curve. Then select the second.
- 3 Erase the original curve if necessary.

Command line: RULESURF

To create a tabulated mesh

- 1 Click Draw menu > Modeling > Meshes > Tabulated Mesh.
- 2 Specify a path curve.
- 3 Specify a direction vector.
- 4 Erase the original objects if necessary.

Command line: TABSURF

To create a revolved mesh

- 1 Click Draw menu > Modeling > Meshes > Revolved Mesh.
- 2 Specify a path curve.

The path curve, which defines the *N* direction of the mesh, can be a line, arc, circle, ellipse, elliptical arc, 2D polyline, 3D polyline, or spline. If you select a circle, closed ellipse, or closed polyline, the mesh is closed in the *N* direction.
- 3 Specify the axis of revolution.

The direction vector can be a line or an open 2D or 3D polyline. If you choose a polyline, the vector sets the rotation axis from its first vertex to its last vertex. Any intermediate vertices are ignored. The axis of revolution determines the *M* direction of the mesh.
- 4 Specify the start angle. Then specify the included angle.

If you specify a nonzero start angle, the mesh is generated at a position offset from the path curve by that angle. The included angle specifies how far about the axis of revolution the mesh should extend.
- 5 Erase the original objects if necessary.

Command line: REV SURF

To create an edge-defined Coons surface patch mesh

- 1 Click Draw menu > Modeling > Meshes > Edge Mesh.
- 2 Select the four edges in any order.

The first edge you select determines the mesh's *M* direction.

Command line: EDGESURF

Create Wireframe Models

A wireframe model is an edge or skeletal representation of a real-world 3D object using lines and curves.

You can use a wireframe model to

- View the model from any vantage point
- Generate standard orthographic and auxiliary views automatically
- Generate exploded and perspective views easily
- Analyze spatial relationships, including the shortest distance between corners and edges, and checking for interferences
- Reduce the number of prototypes required

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 the XEDGES command to create wireframe geometry from regions and 3D solids and surfaces. XEDGES extracts all the edges on the selected objects or subobjects.

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.

Methods for Creating Wireframe Models

You can create wireframe models by positioning any 2D planar object anywhere in 3D space, using the following methods:

- Entering 3D coordinates. You enter coordinates that define the *X*, *Y*, and *Z* location of the object.
- Setting the default workplane (the *XY* plane of the UCS) on which you will draw the object.
- Moving or copying the object to its proper 3D location after you create it.

Wireframe modeling is a skill that requires practice and experience. The best approach to learning how to create wireframe models is to begin with simple models before attempting models that are more complex.

To drop a perpendicular line from a 3D point down to the *XY* plane

- 1 Click Draw menu ▶ Line.
- 2 Use an object snap to specify a point on an object that is not on the *XY* plane of the UCS.
This defines the first point of the line.
- 3 Enter **.xy** and then enter **@** at the Of prompt.
This operation extracts the *X* and *Y* coordinate values from the first point.
- 4 Enter **0** to specify the *Z* value.
Using coordinate filters, the *X* and *Y* values extracted from the first point are combined with a new *Z* value (0) to finish the definition of the second point.
- 5 Press ENTER to end the command.

To create wireframe geometry by extracting edges

- 1 Click Modify menu ▶ 3D Operations ▶ Extract Edges.
- 2 Select any combination of the following objects:
 - Solids
 - Surfaces

- Regions
- Edges (on 3D solids or surfaces)
- Faces (on 3D solids or surfaces)

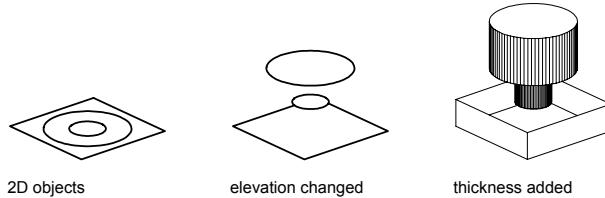
3 Press ENTER.

Command line: XEDGES

Add 3D Thickness to Objects

Thickness is a property of certain objects that gives them a 3D appearance.

The 3D thickness of an object is the distance that object is extended, or thickened, above or below its location in space. Positive thickness extrudes upward in the positive Z direction; negative thickness extrudes downward (negative Z). Zero (0) thickness means that there is no 3D thickening of the object. The Z direction is determined by the orientation of the UCS at the time the object was created. Objects with a non-zero thickness can be shaded and can hide other objects behind them.



The thickness property changes the appearance of the following types of objects:

- 2D solids
- Arcs
- Circles
- Lines
- Polylines (including spline-fit polylines, rectangles, polygons, boundaries, and donuts)
- Text (only if created as a single-line text object using an SHX font)

- Traces

- Points

Modifying the thickness property of other types of objects does not affect their appearance.

You can set the default thickness property for new objects you create by setting the THICKNESS system variable. Change the thickness property of an existing object using the Properties palette. The 3D thickening is applied uniformly on an object; a single object cannot have different thicknesses for its various points.

You may need to change the 3D viewpoint to see the effect of thickness on an object.

See also:

[Create Solids and Surfaces from Objects](#)

To set the 3D thickness of new objects

- 1 Click Format menu ► Thickness.
- 2 On the command line, enter the value for the thickness distance.
New objects are created with the specified 3D thickness.

Command line: THICKNESS

To change the 3D thickness of existing objects

- 1 Select the objects whose 3D thickness you want to change.
- 2 Right-click one of the objects. Click Properties.
- 3 In the Properties palette, select Thickness and enter a new value.
The selected objects now display the specified 3D thickness.

Standard toolbar



Command line: PROPERTIES

Modify 3D Solids and Surfaces

23

In this chapter

- Manipulate 3D Solids and Surfaces
- Press or Pull Bounded Areas
- Use Grip Tools to Modify Objects
- Add Edges and Faces to Solids
- Separate 3D Solids
- Shell 3D Solids
- Clean and Check 3D Solids

Manipulate 3D Solids and Surfaces

You can use grips or the Properties palette to change the shape and size of solids and surfaces. The type of solid or surface and how it was created determine how it can be manipulated.

Manipulate Individual Solids and Surfaces

You can use grips or the Properties palette to change the size and shape of some individual solids and surfaces.

You can use grips or the Properties palette to change the shape and size of primitive solids while maintaining their original basic form. For example, you can change the height and base radius of a cone, but it remains a cone.

As another example, if you change a pyramid from a four-sided pyramid that ends in a point to an eight-sided pyramid that ends in a planar surface (pyramid frustum), it is still a pyramid.

Extruded Solids and Surfaces

You create extruded solids and surfaces with the EXTRUDE command. When selected, extruded solids and surfaces display grips on their profiles. A *profile* is the original form that is used to create the extruded solid or surface and defines the object's shape. You can use these grips to manipulate the profile of the solid or surface, which changes the shape of the entire solid or surface.

If a path was used for the extrusion, the path is displayed and can be manipulated with grips. If a path wasn't used, a height grip will display at the top of the extruded solid or surface, which allows you to manipulate the object's height.

Polysolids

You can use grips to change the shape and size of polysolids, including the profile of the polysolid. The grips can be moved in the XY plane of the solid. The profile of a polysolid is always angular (rectangular by default).

Swept Solids and Surfaces

Swept solids and surfaces display grips on the swept profile as well as on the sweep path. You can use these grips to modify the solid or surface.



Swept polyline

Polyline vertex stretched
to change profile

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

You can click and drag grips on any of the defining lines or curves to modify the solid or surface. If the lofted solid or surface contains a path, you can only edit the portion of the path that is between the first and last cross sections.



Cross sections
for lofting

Lofted solid

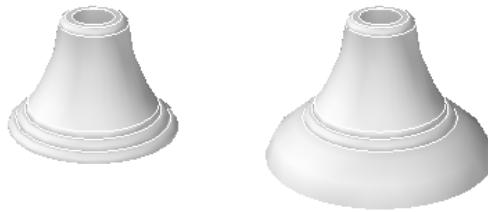
Lofted solid with
modified bottom
cross section

You can also use the Geometry area of the Properties palette to modify the contour of a lofted solid or surface at its cross sections. When you first select a lofted solid or surface, the Geometry area of the Properties palette displays the settings that were specified through the Loft Settings dialog box when the solid or surface was created.

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 or surface.



Revolved
surface

Revolved surface
with modified profile

See also:

[Create 3D Solids and Surfaces](#)

To relocate the axis of revolution for a revolved solid or surface

- 1 In a drawing, select a revolved solid or surface.
- 2 Select the grip on the axis of revolution.
- 3 Click to relocate axis of revolution.

To modify the contour of a lofted solid or surface through the Properties palette

- 1 In a drawing, select a lofted solid or surface that was created with cross sections.
- 2 If the Properties palette is not displayed, click Tools > Palettes > Properties.
- 3 On the Properties palette, Geometry area, change the Surface Normals setting to any of the following:
 - **Ruled.** Specifies that the solid or surface is ruled between the cross sections and has sharp edges at the cross sections.
 - **Smooth.** Specifies that a smooth solid or surface is interpolated between the cross sections and has sharp edges at the cross sections. The tangent direction at the cross sections is controlled by the adjoining cross sections.
 - **First Normal.** Specifies that the surface normal is normal to the first cross section.
 - **Last Normal.** Specifies that the surface normal is normal to the last cross section.

- **Ends Normal.** Specifies that the surface normal is normal to both the first and last cross sections.
- **All Normal.** Specifies that the surface normal is normal to all cross sections.
- **Use Draft Angles.** Controls the draft angle of the first and last cross sections of the lofted solid or surface. The draft angle settings are controlled by the following system variables: LOFTANG1, LOFTANG2, LOFTMAG1, and LOFTMAG2.

Work with Composite Solids

Composite solids are created from two or more individual solids through any of the following commands: UNION, SUBTRACT, and INTERSECT. (FILLET and CHAMFER also create composite solids.) You can manipulate the original individual solids that make up a composite solid or the full form of the composite.

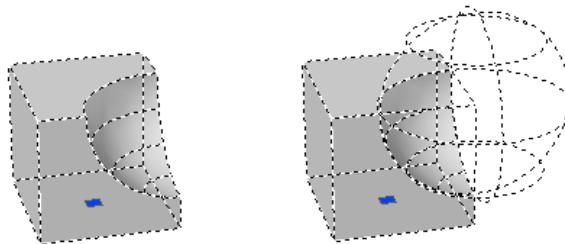
See also:

[Modify Edges on 3D Solids](#)

Display Original Forms of Composite Solids

By default, 3D solids record a *history* of their original forms. This history allows you to see the original forms that make up composite solids.

For composite solids, set the Show History property to Yes to display wireframes of the original forms (in a dimmed state) of the original individual solids that make up the composite.



Composite solid
with Show History
set to No

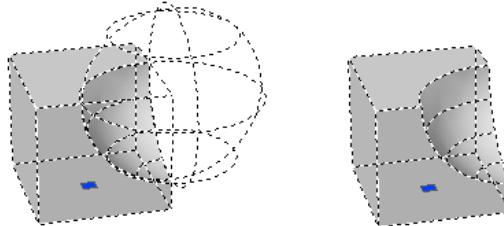
Composite solid
with Show History
set to Yes

Displaying a composite solid's history is useful when modifying composite solids. Grips are displayed on the original forms that make up the composite. For more information about using grips to manipulate composite solids, see Manipulate Composite Solids.

You can use the SHOWHIST system variable to override the Show History property setting in the Properties palette.

In order for a composite solid to record a history of its original parts, the original individual solids must have their History property set to Record. This property is set to Record by default, but can be changed in the Solid History area of the Properties palette or through the SOLIDHIST system variable.

You can remove the history of a composite solid by selecting the solid and changing its History property setting to None. You can also use the BREP command to remove the history from an individual or a composite solid. Once a solid's history has been removed, the solid's original parts cannot be selected and modified. You can then change the solid's History setting back to Record, restarting the solid's history from scratch.



Composite solid with
Show History set to Yes

Composite solid with
history removed and
Show History set to Yes

Removing a composite solid's history can be useful when you are creating very complex composite solids. For example, you might need to create a complex composite solid that will serve as the base for additional composite solids. Once you've completed the base form, you can remove the history. Then you can set the History property back to Record, so the forms of additional objects that you add to the composite can be edited later.

- 1 If the Properties palette is not displayed, click Tools ► Palettes ► Properties.
- 2 In a drawing, select a 3D composite solid.

- 3 On the Properties palette, Solid History area, under Show History, select Yes.
 - 1 If the Properties palette is not displayed, click Tools > Palettes > Properties.
 - 2 In a drawing, select a 3D solid.
 - 3 On the Properties palette, Solid History area, under History, select None.

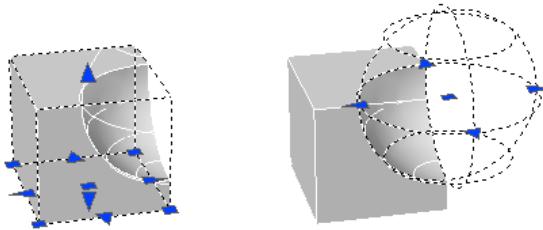
Command line: BREP

- 1 If the Properties palette is not displayed, click Tools > Palettes > Properties.
- 2 In a drawing, select a solid.
- 3 On the Properties palette, Solid History area, under History, select Record.

Manipulate Composite Solids

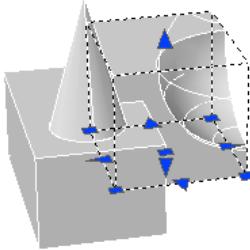
You can manipulate the entire form of a composite solid or the original forms that make up the composite.

You can select a composite solid in a drawing and move, scale, or rotate the solid using grip modes or grip tools.



If the selected individual form does not contain its history, you can move, rotate, scale, or delete the form.

A composite solid may be made up of other composite solids. You can select the original individual forms of composites within composites by holding down the CTRL key and continuing to click on the forms.



You can also change the size and shape of composite solids by clicking and dragging grips on individual faces, edges, and vertices. For more information, see [Select and Modify 3D Subobjects](#).

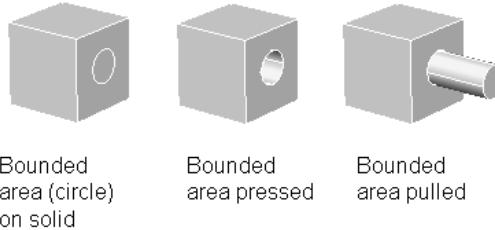
See also:

[Use Grip Modes](#)
[Make Multiple Copies with Grips](#)
[Use Grip Tools to Modify Objects](#)

- 1 Press and hold CTRL.
- 2 Click on an individual solid that is part of a composite solid.
- 3 Repeat step 2 until you've selected the required form.

Press or Pull Bounded Areas

You can press or pull bounded areas by pressing and holding CTRL + ALT, and then picking the area. The area must be bounded by coplanar lines or edges.



You can press or pull bounded areas that are defined by the following types of objects:

- Any area that can be hatched by picking a point (with zero gap tolerance)

- Areas enclosed by crossing coplanar, linear geometry, including edges and geometry in blocks
- Closed polylines, regions, 3D faces, and 2D solids that consist of coplanar vertices
- Areas created by geometry (including edges on faces) drawn coplanar to any face of a 3D solid

Click the Play arrow to start the animation.

When you press or pull a bounded area, you cannot taper it. You can, however, select the edges of the bounded area after it's been pressed or pulled and manipulate the edges to achieve the same effect.

Through the IMPLIEDFACE system variable, you can specify whether or not bounded area can be detected and pressed or pulled.

Modeling toolbar



Command line: **presspull**

3D Make panel, Press-pull

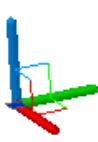
Use Grip Tools to Modify Objects

You can use grip tools to move and rotate objects and subobjects in a 3D view. Grip tools are displayed when you select objects and subobjects in a 3D view with a 3D visual style applied or when you use the 3DMOVE and 3DRotate commands.

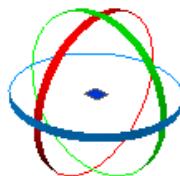
Understand Grip Tools

Grip tools are icons 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.

There are two types of grip tools: the move grip tool and the rotate grip tool.



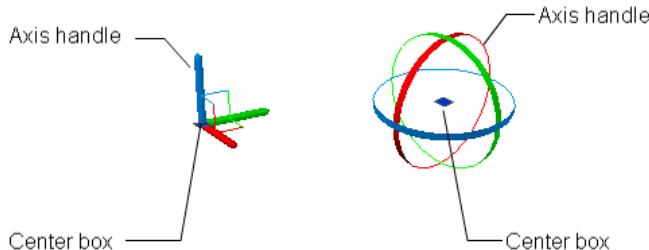
Move grip tool



Rotate grip tool

You display grip tools with the 3DMOVE or 3DROTATE commands or by clicking the 3D Move or 3D Rotate buttons on the dashboard.

A grip tool can be placed anywhere in 3D space. This location (indicated by the center box [or base grip] of the grip tool) sets the base point for the movement or rotation. This is equivalent to temporarily changing the position of the UCS while you move or rotate the selected objects. You then use the lines (axis handles) on the grip tool to constrain the movement or rotation to an axis or plane.



When you select objects first, and then click an axis handle and start moving the selected objects, you can press SPACEBAR to change to the rotate grip tool. You can press SPACEBAR again to change back to the move grip tool. When you change grip tools this way, the movement remains constrained to the selected axis.

When you start the 3DMOVE or 3DROTATE command before selecting objects, the grip tool is attached to the cursor after you've created your selection set. You can then click to place the grip tool anywhere in 3D space. You can relocate a grip tool by clicking its center box (base grip) and then clicking to specify a new location.

Grip tools are only displayed in a 3D view with a 3D visual style applied. If you have the 2D Wireframe visual style applied, 3DMOVE and 3DROTATE will automatically change the visual style to 3D Wireframe.

By default, when you select objects first and then start the 3DMOVE or 3DROTATE command to display a grip tool, the move grip tool displays and is located at the same location as the UCS and aligned with it. You can use

the GTLOCATION system variable to set the default initial location for grip tools.

You use the GTAUTO system variable to specify whether or not grip tools display automatically when selecting objects before starting a command in a 3D view set to a 3D visual style.

The GTDEFAULT system variable controls whether or not the 3DMOVE and 3DROTATE commands start automatically when the MOVE and ROTATE commands (respectively) are started in a 3D view. By default, the 3DMOVE and 3DROTATE commands do not start automatically when the MOVE and ROTATE commands (respectively) are started in a 3D view.

To set the default initial location for grip tools

- 1 On the command line, enter **gtlocation**.
- 2 Do one of the following:
 - Enter **0** to set the location as the same location as the UCS.
 - Enter **1** to set the location to the last selected object or subobject.
- 3 Press ENTER.

To relocate a grip tool

- 1 Click the center box (base grip) of the grip tool.
- 2 Click to specify a new location.

To change the type of grip tool while moving or rotating objects

- 1 On the dashboard, 3D Make control panel, click 3D Move or 3D Rotate.
- 2 Select the objects and subobjects you want to move or rotate.
Press and hold CTRL to select subobjects (faces, edges, and vertices).
Release CTRL to select objects.
- 3 When you're done selecting objects, press ENTER.
The move or rotate grip tool is displayed attached to the cursor.
- 4 Click to place the grip tool, specifying the base point for the operation.
- 5 Hover the cursor over an axis handle on the grip tool until it turns yellow and the vector displays, and then click the axis handle.
- 6 Press SPACEBAR.

You can press SPACEBAR again to change back to the other grip tool. When you change grip tools this way, the movement remains constrained to the selected axis.

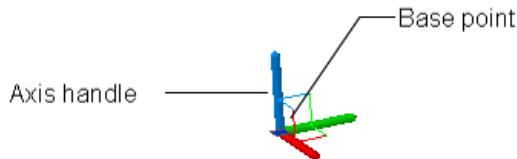
Modeling toolbar

Command line: 3DMOVE or 3DROTATE

Use the Move Grip Tool to Modify Objects

The move grip tool allows you to move a selection set of objects and subobjects freely or constrain the movement to an axis or plane.

After you've selected the objects and subobjects that you want to move, you place the grip tool anywhere in 3D space. This location (indicated by the center box [or base grip] of the grip tool) sets the base point for the movement and temporarily changes the position of the UCS while you move the selected objects.

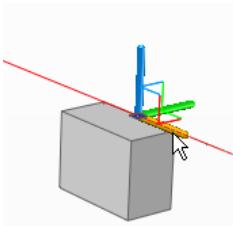


You then move the objects freely by dragging outside the grip tool or specify the axis or plane to which you will constrain the movement.

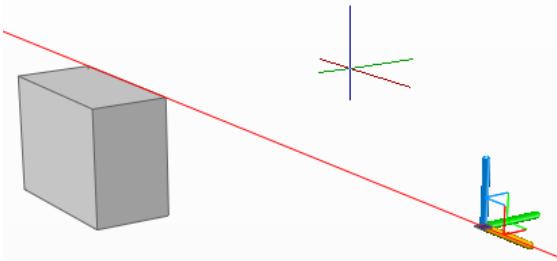
NOTE By default, when you create a selection set in a 3D view with a 3D visual style applied before you start a command, the move grip tool automatically displays. You can specify that the move grip tool does not automatically display by setting the GTAUTO system variable to 0.

Constrain the Movement to an Axis

You can use the move grip tool to constrain the movement to an axis. You hover the cursor over an axis handle on the grip tool until the vector displays aligned with the axis, and then click the axis handle.

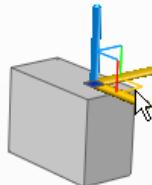


Now when you drag the cursor, the selected objects and subobjects will move only along the specified axis. You can click or enter a value to specify the distance of the move from the base point.

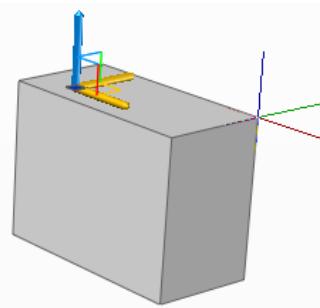


Constrain the Movement to a Plane

You can use the move grip tool to constrain the movement to a plane. You hover the cursor over the point where the two lines coming off the axis handles (that determine the plane) meet until the lines turn yellow, and then click that point.

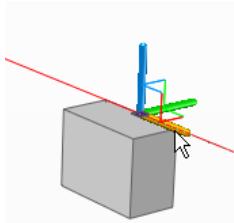


Now when you drag the cursor, the selected objects and subobjects will move only along the specified plane. You can click or enter a value to specify the distance of the move from the base point.



To move objects in 3D space along a specified axis

- 1 On the dashboard, 3D Make control panel, click 3D Move.
- 2 Select the objects and subobjects you want to move.
Press and hold CTRL to select subobjects (faces, edges, and vertices).
Release CTRL to select objects.
- 3 When you're done selecting objects, press ENTER.
The move grip tool is displayed attached to the cursor.
- 4 Click to place the move grip tool, specifying the base point for the move.
- 5 Hover the cursor over an axis handle on the grip tool until it turns yellow
and the vector displays, and then click the axis handle.



- 6 Click or enter a value to specify the distance of the move.

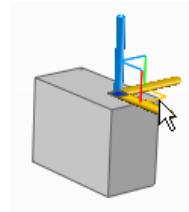
Modeling toolbar



Command line: 3DMOVE

To move objects in 3D space constrained to a specified plane

- 1 On the dashboard, 3D Make control panel, click 3D Move.
- 2 Select the objects and subobjects you want to move.
Press and hold CTRL to select subobjects (faces, edges, and vertices). Release CTRL to select objects.
- 3 When you're done selecting objects, press ENTER.
The move grip tool is displayed attached to the cursor.
- 4 Click to place the move grip tool, specifying the base point for the move.
- 5 Hover the cursor over the point where the two lines coming off the axis handles (that determine the plane) meet until the lines turn yellow, and then click.



- 6 Click or enter a value to specify the distance of the move.

Modeling toolbar

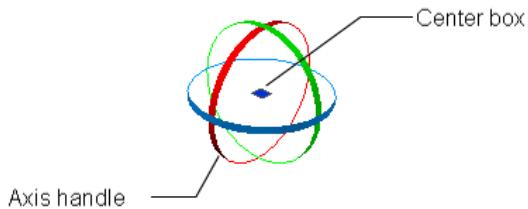


Command line: 3DMOVE

Use the Rotate Grip Tool to Modify Objects

The rotate grip tool allows you to rotate objects and subobjects freely or constrain the rotation to an axis.

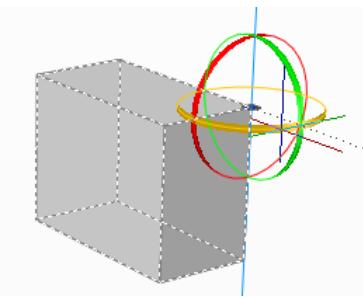
After you've selected the objects and subobjects that you want to rotate, you place the grip tool anywhere in 3D space. This location (indicated by the center box [or base grip] of the grip tool) sets the base point for the movement and temporarily changes the position of the UCS while you rotate the selected objects.



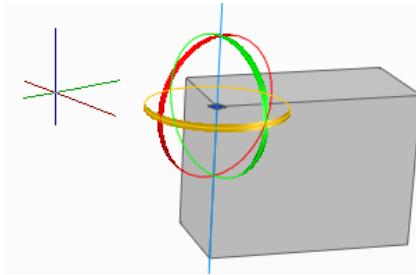
You then rotate the objects freely by dragging outside the grip tool or specify the axis to which you will constrain the rotation.

Constrain the Rotation to an Axis

You can use the rotate grip tool to constrain the rotation to an axis. You hover the cursor over one of the axis handles on the grip tool until it turns yellow, and the yellow vector displays aligned with the axis. Then you click the axis line.



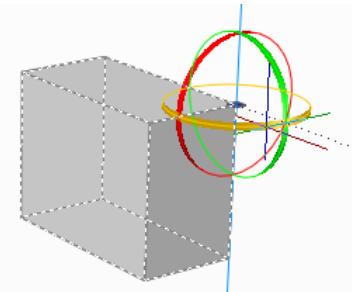
Now when you drag the cursor, the selected objects and subobjects will rotate about the base point along the specified axis. You can click or enter a value to specify the angle of the rotation.



To rotate objects in 3D space along a specified axis

- 1 On the dashboard, 3D Make control panel, click 3D Rotate.

- 2** Select the objects and subobjects you want to rotate.
Press and hold CTRL to select subobjects (faces, edges, and vertices).
Release CTRL to select objects.
- 3** When you're done selecting objects, press ENTER.
The rotate grip tool is displayed attached to the cursor.
- 4** Click to place the rotate grip tool, specifying the base point for the move.
- 5** Hover the cursor over an axis handle on the grip tool until it turns yellow and the vector displays, and then click.



- 6** Click or enter a value to specify the angle of the rotation.

Modeling toolbar



Command line: 3DROTATE

Add Edges and Faces to Solids

You can modify the appearance of a face on a 3D solid by imprinting it with an object that intersects the selected face. Imprinting combines the object with the face, creating an edge.

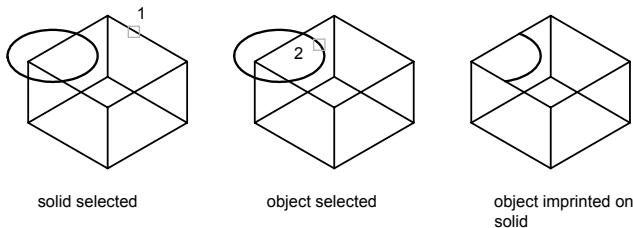
With the IMPRINT command, you can create new faces on 3D solids by imprinting arcs, circles, lines, 2D and 3D polylines, ellipses, splines, regions, bodies, and 3D solids. For example, if a circle intersects with a 3D solid, you can imprint the intersecting curves on the solid. You can delete or retain the original imprinted objects to be used for further editing. The imprinted object must intersect a face or faces on the selected solid in order for the imprinting to be successful.

In some cases, it is not possible to move, rotate, or scale the following subobjects, and imprinted edges and faces may get lost when you move, rotate, or scale the following subobjects:

- Faces with imprinted edges or faces
- Edges or vertices with adjacent faces that contain imprinted edges or faces

To imprint a 3D solid object

- 1 Click Modify menu > Solid Editing > Imprint Edges.
- 2 Select the 3D solid object (1).
- 3 Select the object that you want to imprint (2).
- 4 Press ENTER to retain the original objects, or enter **y** to delete them.
- 5 Select additional objects to imprint or press ENTER.
- 6 Press ENTER to complete the command.



Solid Editing toolbar



Command line: IMPRINT

3D Make panel, Imprint

Separate 3D Solids

You can disassemble a 3D solid object into its original component 3D solids.

You can separate (take apart) composite solids. The composite 3D solid object cannot share a common area or volume. After separation of the 3D solid, the

individual solids retain the layers and colors of the original. All nested 3D solid objects are separated into their simplest form.

To separate a 3D solid composite into individual solids

- 1 Click Modify menu ► Solid Editing ► Separate.
- 2 Select the 3D solid object.
- 3 Press ENTER to complete the command.

Solid Editing toolbar



Command line: SOLIDEDIT

Shell 3D Solids

You can create a shell (a hollow, thin wall with a specified thickness) from your 3D solid object.

New faces are created by offsetting existing ones inside or outside their original positions. Continuously tangent faces are treated as single faces when offsetting.

To create a 3D solid shell

- 1 Click Modify menu ► Solid Editing ► Shell.
 - 2 Select the 3D solid object.
 - 3 Select a face to be excluded from shelling (1).
 - 4 Select additional faces to be excluded or press ENTER.
 - 5 Specify the shell offset value.
- A positive offset value creates a shell in the positive face direction; a negative value creates a shell in the negative face direction.
- 6 Press ENTER to complete the command.



Solid Editing toolbar



Command line: SOLIDEDIT

Clean and Check 3D Solids

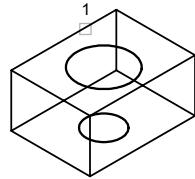
You can use the SOLIDEDIT command to remove redundant faces, edges, and vertices from a 3D solid automatically, and check to make sure that the 3D solid is valid.

You can remove edges or vertices if they share the same surface or vertex definition on either side of the edge or vertex. The body, faces, or edges on the solid object are checked, and adjacent faces that share the same surface are merged. All redundant edges, imprinted as well as unused, on your 3D solid object are deleted.

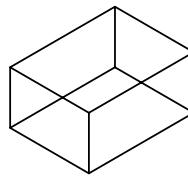
You can check to see if your solid object is a valid 3D solid object. With a valid 3D solid, you can modify the object without incurring failure error messages. If the 3D solid is not valid, you cannot edit the object.

To clean up a 3D solid object

- 1 Click Modify menu ▶ Solid Editing ▶ Clean.
- 2 Select the 3D solid object (1).
- 3 Press ENTER to complete the command.



solid selected



solid cleaned

Solid Editing toolbar



Command line: SOLIDEDIT

To validate a 3D solid object

- 1 Click Modify menu ► Solid Editing ► Check.
- 2 Select the 3D solid object.
- 3 Press ENTER to complete the command.

Solid Editing toolbar



Command line: SOLIDEDIT

Create Sections and 2D Drawings from 3D Models

24

In this chapter

- Section 3D Solids
- Work with Section Objects
- Live Sectioning
- Generate 2D and 3D Sections
- Create a Flattened View

Section 3D Solids

You can create a cross section through a 3D solid. The result can be a two-dimensional object that represents the shape of the section. You also have the option of using a cutting plane, called a *section object* that lets you see the cut profile of intersected solids in real-time.

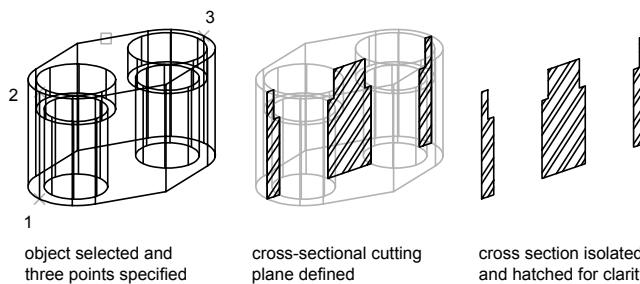
With the SECTION command, you can create a cross section through a solid. You specify three points to define the plane of the cross section. You can also define the cross-sectional plane with another object, the current view, the Z axis, or the XY, YZ, or ZX plane. The cross-sectional plane is placed on the current layer.

With the SECTIONPLANE command, you can create a *section object* that acts as a cutting plane through solids, surfaces, or regions (two-dimensional areas created from closed shapes or loops). If you turn on *live sectioning*, moving the section object throughout the 3D model in model space reveals inner details in real-time.

To create a section object, you move the cursor over any face of a 3D model and click to automatically place the section object. You can also pick points to create a straight cutting plane, or pick multiple points to create one with jogged segments. Another method is to specify an orthographic view, such as front, top, or back.

To create a cross section of a solid

- 1 Click Draw menu ▶ Modeling ▶ Section.
- 2 Select the object to cross section.
- 3 Specify three points to define the cross-sectional plane.



Command line: SECTION

NOTE If you are applying hatching to the cross-sectional cutting plane, you must align the UCS with the cross-sectional cutting plane first.

Work with Section Objects

You can create sectional views from 3D models based on where a transparent cutting plane called a *section object* intersects the model. When *live sectioning* is active on a section object, you can see the inner details of the model, when the section plane is stationary or as you move it through the 3D model. Live sectioning allows you to dynamically change the cut profile of the intersected solid.

Overview of Section Objects

No procedures for this topic.

Create Section Objects

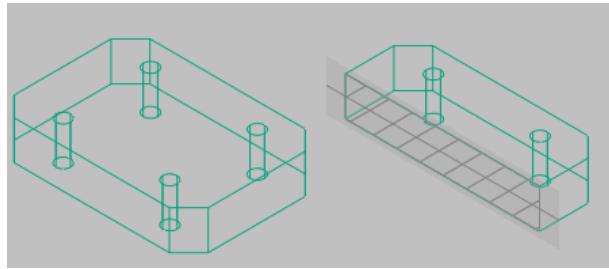
You can create section objects using the SECTIONPLANE command.

The default method is to move the cursor over the face of your 3D model and then click to place the section object. The section plane is automatically aligned to the plane of the face you selected.

You can also create a section object by selecting two points to create a straight section line.

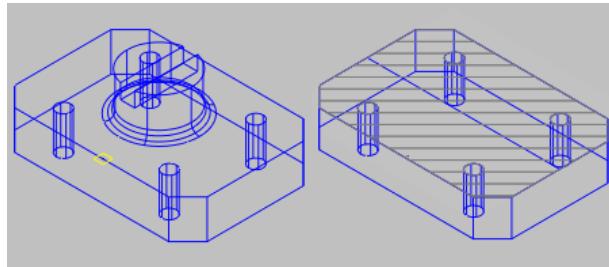
If you need a section line that has jogged segments, the Draw Section option lets you pick multiple points throughout the 3D model.

Using the Orthographic option, you can quickly create a section object that is aligned to a preselected orthographic plane. Before the section object is created, all the 3D objects are evaluated in the drawing and an imaginary 3D boundary around their extents is calculated. The section object is then positioned to pass through the center of the 3D boundary and oriented to the selected orthographic option you choose.



To create a section object by selecting a face

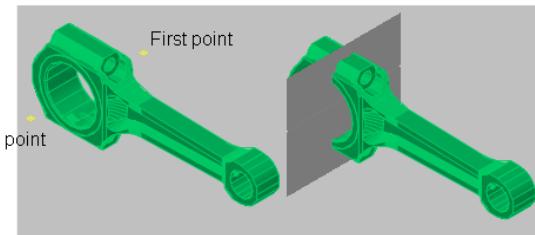
- 1 Click Draw menu » Modeling » Section Plane.
- 2 Click to select a face on your model.
A section object is created on the plane of the selected face.
- 3 Click the section line to display its grips.
- 4 Select a grip to move the section plane through the 3D object.
A section object is created in the Section Plane state. Live sectioning is turned on.



To create a section object using two points

- 1 Click Draw menu » Modeling » Section Plane.
- 2 Specify the first point of the section object.
- 3 Specify the endpoint.

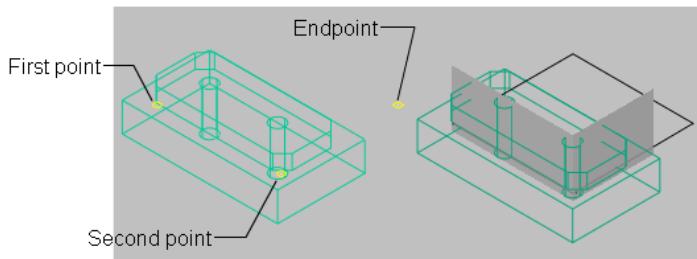
The section object is created between the two points. Live sectioning is turned off.



To create a section object with jogged segments

- 1 Click Draw menu » Modeling » Section Plane.
- 2 Enter **d** (Draw Section).
- 3 Specify the start point of the section object.
- 4 Specify a second point to create the first jogged segment.
From this point, you cannot create segments that intersect.
- 5 Continue specifying segment endpoints, then press ENTER.
- 6 Specify a point in the direction of the sectional cut.

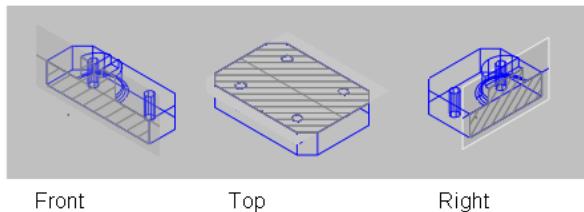
A section object is created that has multiple segments in a Section Boundary state. Live sectioning is turned off.



To create a section object on a preset orthographic plane

- 1 Click Draw menu » Modeling » Section Plane.
- 2 Enter **o** (Orthographic).
- 3 Select an alignment option.

A section object is created that is centered in an imaginary 3D boundary around all the 3D objects in the drawing. It is placed on the selected orthographic plane. Live sectioning is turned on.



Set Section Object States

Section objects have three states: Section Plane, Section Boundary, and Section Volume. Depending on the state you choose, you can view the cutting plane as a 2D plane, a 2D box, or a 3D box. Grips allow you to make adjustments to the length, width, and height of the cutting area.

- **Section Plane.** The section line and transparent section plane indicator display. 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.

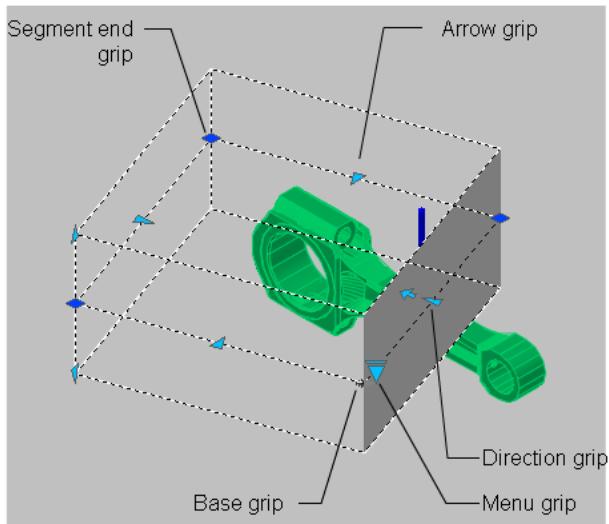
To change a section object's state using the Menu grip

- 1 Select the section object to display its grips.
- 2 Click the Menu grip.
- 3 On the section state menu, click the state you want.

The section object is updated in the viewport to the selected state.

Use Section Object Grips

When you select a section object, various types of grips display that perform different functions. These grips are



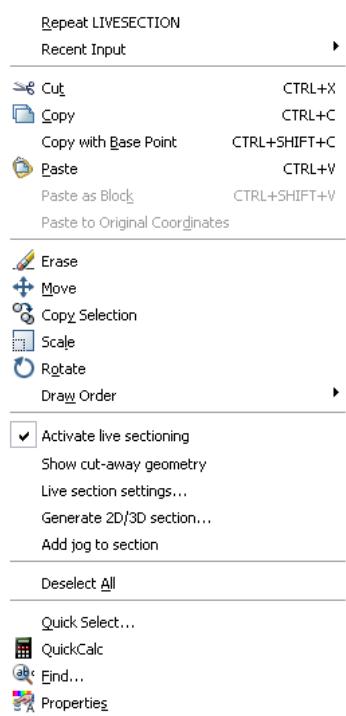
- **Base grip.** Acts as the basepoint grip for the section object for moving, scaling, and rotating the section object. It is always adjacent to the Menu grip.
- **Menu grip**. Displays a menu of section object states.
- **Direction grip.** Shows the direction from which 2D sections are taken and from which direction live sectioning will be viewed. To reverse the direction of the section plane, click the Direction grip.
- **Arrow grip**. Moves segments of the section line perpendicular to themselves. As segments are moved, the length of the segments are changed but the angle of adjacent segments remains the same.
- **Second grip.** Rotates the section object around the base grip.
- **Segment end grips.** Behave like polyline grips, but they can't be moved so that segments intersect. Segment end grips display at the end points of jogged segments.

NOTE Only one section object grip can be selected at a time.

No procedures for this topic.

Access Section Object Shortcut Menu

With a preselected section object, you can access the following controls on the shortcut menu by right-clicking the section line.



- **Activate Live Sectioning.** Turns live sectioning on and off for the selected section object.
- **Show Cut-away Geometry.** Displays geometry that has been cut using the display settings from the Section Settings dialog box. This option is available when live sectioning is turned on.
- **Live Section Settings.** Displays the Section Settings dialog box.
- **Generate 2D/3D Section.** Displays the Generate Section/Elevation dialog box.
- **Add Jog to Section.** Adds an additional segment, a jog, to the section line.

No procedures for this topic.

Change Section Object Properties

Section objects have properties just as other AutoCAD objects do. Properties are stored in the section line and can be accessed in the Properties palette. For example, you can change the section object's name, layer and linetype, and the color and transparency of the section plane indicator (the section object's transparent cutting plane).

To rename a section object

- 1 Right-click the section object. Click Properties.
- 2 Under Section Object, in Name, enter a different name.

To change a section object's state using the Properties palette

- 1 Right-click the section object. Click Properties.
- 2 Under Section Object, change the value for Type.

The section object is updated in the viewport to the selected state.

To change the transparency and color of the section plane indicator

- 1 Right-click the section object. Click Properties.
- 2 Under Section Object, enter a value from 1 - 100 for Plane Transparency. Setting it to 1 makes the section plane indicator opaque.
- 3 Select a color for Plane Color.

The section plane indicator is updated in the viewport.

To change the height of the section plane indicator

- 1 Right-click the section object. Click Properties.
- 2 Under Geometry, change the value for Top Plane. This changes the distance from the section line to the upper edge of the section plane.
- 3 Change the value for Bottom Plane. This changes the distance from the section line to the bottom edge of the section plane.

The section plane indicator is updated in the viewport.

Associate Section Objects with Views and Cameras

Section objects can be associated with named views and cameras in the View Manager.

When a named view or camera that has a section object associated with it is made active, live sectioning is turned on for that section object. For a 3D model that has multiple section objects, it is convenient to associate a particular section object to a view or camera. Later, you can quickly restore a saved sectional view or camera, which automatically activates live sectioning for the associated section object.

For example, you've 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. You also want to view the sectional cut to be perpendicular to your line of sight. By associating each section object with a view or camera, you can quickly switch between the two views and see the desired cross section.

To associate a section object with a view or camera

- 1 Click View menu » Named Views.
- 2 In the View Manager, in the left pane, under Model Views, select a named view or camera.
- 3 In the General section, in the Live Section drop-down list, select a section object by name.
- 4 Click OK.

The section object is now associated with a named view or camera. When the named viewed or camera is restored, live sectioning is turned on for the associated section object.

Publish Section Objects

You can control the visibility of section objects when you render, plot, or view them in the Autodesk DWF Viewer.

Render Section Objects

With live sectioning turned on, all lines on a section object render as 2D lines. The section plane indicator renders as a transparent material. Its degree of transparency is controlled in the Properties palette of the section object.

Plot Section Objects

When a section object is in a Section Boundary or Section Volume state, displayed lines do not plot. The section plane indicator plots as if it was assigned a transparent material, but not with the same fidelity as when rendered. If you do not want to plot the section line, place the section object on a layer that is turned off.

View Section Objects in the Autodesk DWF Viewer

When live sectioning is active on a section object, the 3D model is displayed in the DWF Viewer with the best possible visual fidelity. Geometry that is hidden by live sectioning is also hidden when displayed in the DWF Viewer. Display settings assigned for live sections are displayed in the viewer as they are in the drawing. For example, dashed linetypes and a hatch pattern assigned to the intersected area of a 3D model will be represented as such in the DWF Viewer.

The section object is not visible in the DWF Viewer.

No procedures for this topic.

Save Section Object as Tool Palette Tool

Like other objects, you can create tools from individual section objects that can be accessed from a tool palette.

You can have multiple section objects, each having their own set of properties. By saving each section object as a tool, you can then quickly create section objects without having to modify their settings each time.

To save a section object tool as a tool palette tool

- 1** On a section object, select the section line.
- 2** Drag the section object to a tool palette and, without releasing the mouse button, move the cursor to the place on the tool palette where you want the tool.

The black line indicates where the tool will be located.

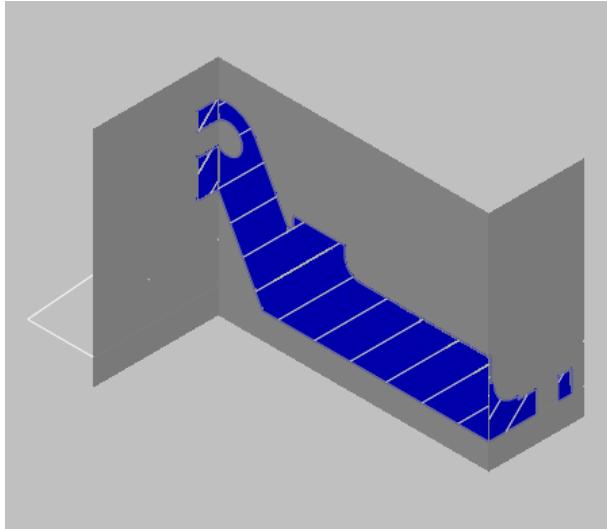
- 3** Release the mouse button.

The section object is saved as a tool on the tool palette.

Add Jogs to a Section

You can create a section line that has multiple segments (jogs), using the Draw Section option of the SECTIONPLANE command. You can also add a jog to

an existing section object using the Add Jog to Section option on the shortcut menu.



Section object created with a jogged segment

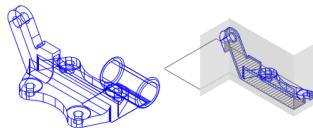
When adding a jog to an existing section object, a segment is created that is perpendicular to the selected segment and in the direction of the Direction grip. Jogs cannot be added to the side or back lines of the section object. The Nearest osnap is temporarily turned on when adding jogs to a section.

After adding jogs, you can refine the jogged sections by using the section object's grips.

To add jogs to a section

- 1 On a section object, select the section line.
- 2 Right-click the line. Click Add Jog to Section.
- 3 Move the cursor over the section line.
- 4 Select a point on the section line where you want to place a jog that is perpendicular to the selected segment.

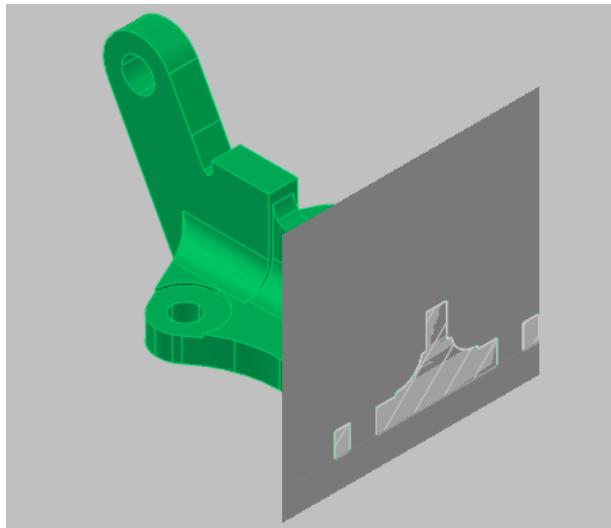
To create additional jogs, repeat the steps.



NOTE The program prevents you from entering jogs that cause the line to intersect itself or close. When this occurs, the entry point is ignored and an error message is displayed on the command line.

Live Sectioning

When a section object has live sectioning turned on, you can dynamically section a 3D model in model space. As you move the section object through the 3D model, its transparent section plane indicator cuts the solid, surface, or region in real-time.



Understand Live Sectioning Behavior

Live sectioning is an analytical tool that allows you to view cut geometry in a 3D solid, surface, or region where the section object intersects the 3D geometry.

You can also use live sectioning to analyze a model by moving the section object back and forth. For example, sliding the section object through an engine assembly can help you visualize its internal components.

Before you begin working with live sectioning, there are some key aspects of the feature you should understand.

- Live sectioning functions only on 3D objects and regions in model space.
- Live sectioning is turned on or off, depending on the option you use when creating the section object. For example, selecting a face creates a section object with live sectioning turned on; the Draw Section option creates a section object with it turned off. After the section object is created, live sectioning for each section object can be manually turned on or off.
- You can have multiple section objects in a drawing; however, live sectioning can only be active for one section object at a time. For example, 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's layer does not turn off live sectioning, but freezing the layer does.
- With live sectioning on, using grips, you can slide segments of the section object to see the effects in the 3D model.

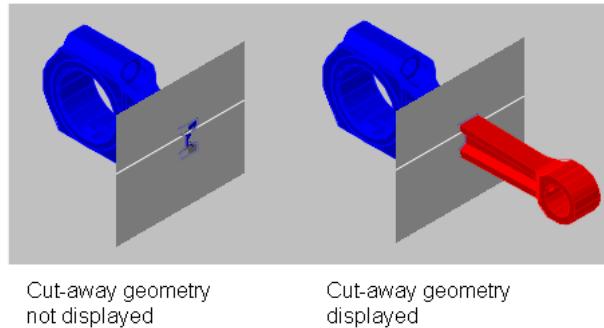
To turn live sectioning on and off

- 1 Select a section object.
- 2 Right-click the section line.
- 3 Click Activate Live Sectioning to turn it on. If on, click to turn it off.

To display cutaway geometry

- 1 Select a section object.
- 2 Right-click the section line. Click Show Cut-away Geometry to turn it on.

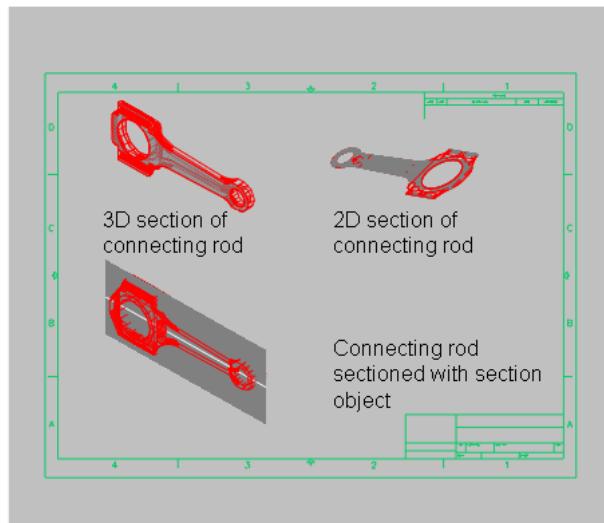
The cut geometry displays according to the Foreground Line settings in the Section Settings dialog box.



Generate 2D and 3D Sections

You can generate a 2D or 3D block representation of the cross-sectional area where a section object intersects a 3D model.

Working from the Generate Section / Elevation Dialog Box, you can choose the type of block that is created. If 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. If you're creating a presentation and need to publish or render a cutaway of the 3D model, you would use the 3D Section option.



Here are some additional considerations for generating sections.

- Both 2D and 3D sections can be inserted into the drawing as an unnamed block or saved to an external file as a wblock. Generated sections are created as blocks that can be renamed and edited using BEDIT.
- Blocks can be rescaled and rotated prior to insertion in the drawing. The insertion basepoint can also be changed.
- 3D objects inserted as xrefs and blocks in the drawing can be used to generate sections.
- 2D sections are created using 2D lines, arcs, circles, ellipses, splines, and hatch patterns.
- 3D sections are created using 3D solids and surfaces, but 2D lines are used for profile outlines and hatch patterns.
- Display properties of 2D section/elevation blocks and 3D section blocks are controlled in the Section Settings Dialog Box.

To generate a 2D or 3D section

- 1 Select the section object. Right-click the section line and click Generate 2D/3D Section.
- 2 In the Generate Section/Elevation dialog box, click 2D Section/Elevation or 3D Section.
- 3 Click Include All Objects.
- 4 Under Destination, click Insert as New Block.
- 5 Click Create.
- 6 Specify an insertion point on the screen.

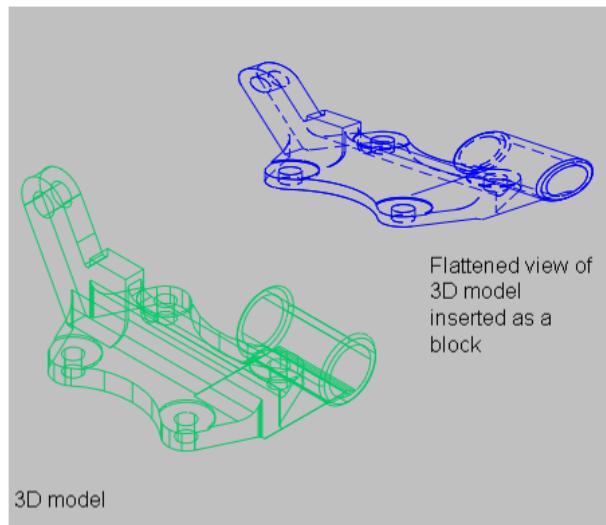
An unnamed block is inserted consisting of 2D or 3D geometry.

Create a Flattened View

With the FLATSHOT command, you can create a flattened view of all the 3D solids and regions in the current view.

The resulting view is a block that is a flattened representation of the 3D model and is projected onto the XY plane. The process is similar to taking a “snapshot” of the entire 3D model with a camera and then laying the photograph flat. After inserting the block, it can be modified because the flattened view is comprised of 2D geometry. This feature is particularly useful for creating technical illustrations.

You're not limited to setting up a particular view before using FLATSHOT. You can set up the drawing in an orthographic or parallel view.



A flattened view is generated with the following parameters:

- All 3D objects in the model space viewport are captured. Place objects you don't want captured on layers that are turned off or frozen.
- Flattened views are created as blocks that can be renamed and edited using BEDIT.
- The generated block is based on the display settings for Foreground and Obscured Lines in the Flatshot dialog box.
- Hidden lines are captured and displayed in the block using the display settings for Obscured Lines in the Flatshot dialog box.
- 3D objects that have been sectioned by section objects are captured in their entirety. FLATSHOT captures these objects as if they were not sectioned.

NOTE To create profile images of 3D solids in paper space, use the SOLPROF command.

To create a flattened 2D view of a 3D model

- 1 Set up the view of the 3D model.
- 2 At the Command prompt, enter **flatshot**.

- 3** In the Flatshot dialog box, under Destination, click one of the options.
- 4** Change the color and linetype settings for Foreground and Obscured lines.
- 5** Click Create.
- 6** Specify an insertion point on the screen to place the block. Adjust the basepoint, scale, and rotation if necessary.

A block is created consisting of 2D geometry that is projected onto the XY plane of the current UCS.

Part 7

Hatches, Notes, Tables, and Dimensions

Chapter 25 Hatches, Fills, and Wipeouts

Chapter 26 Create 3D Models

Chapter 27 Tables

Chapter 28 Dimensions and Tolerances

25

Hatches, Fills, and Wipeouts

See also:

[Modify Complex Objects](#)

In this chapter

- Overview of Hatch Patterns and Fills
- Define Hatch Boundaries
- Choose Hatch Patterns and Solid Fills
- Modify Hatches and Solid-Filled Areas
- Create a Blank Area to Cover Objects

Overview of Hatch Patterns and Fills

Define the Boundaries of a Hatch

You can choose from several methods to specify the boundaries of a hatch.

- Specify a point in an area that is enclosed by objects.
- Select objects that enclose an area.
- Drag a hatch pattern into an enclosed area from a tool palette or DesignCenter.

When you hatch a drawing, whole or partial objects that are not part of the object boundary are ignored.

If a hatch line encounters an object such as text, an attribute, or a solid-fill object, and if the object is selected as part of the boundary set, HATCH hatches around the object.



text object not part
of the boundary set



text object included
in the boundary set

NOTE If you want to hatch an area whose boundary is not quite closed, you can set the HPGAPTOOL system variable to bridge gaps and treat the boundary as closed. HPGAPTOOL applies only to gaps between lines and arcs that, if extended, would meet.

To reduce file size, a hatched area is defined in the drawing database as a single graphical object.

Add Hatch Patterns and Solid Fills

You can use several methods to add hatch patterns to your drawing.

- The HATCH command provides the most options.
- You can drag hatches from a tool palette. Use tool palettes when you need additional speed and convenience.

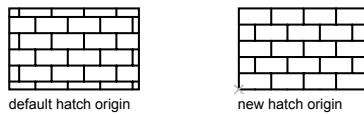
With the Tool Palettes window open, you can right-click a pattern tool to access the Tool Properties dialog box from the shortcut menu. This dialog

box contains several hatch pattern options that are also available through HATCH. For example, you can specify the scale and spacing for the hatch pattern.

- You can also use DesignCenter.

Control the Hatch Origin

By default, hatch patterns always “line up” with each other. However, sometimes you might need to move the starting point, called the *origin point*, of the hatch. For example, if you create a brick pattern, you might want to start with a complete brick in the lower-left corner of the hatched area. In that case, use the Hatch Origin options in the Hatch and Gradient dialog box.



The location and behavior of a hatch pattern depends on the HPORIGIN, HPORIGINMODE, and HPINHERIT system variables, and the location and orientation of the user coordinate system.

Choose a Hatch Pattern

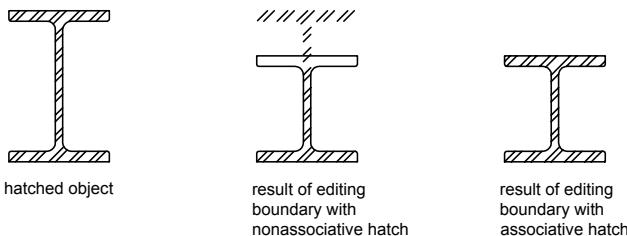
The program supplies a solid fill and more than 50 industry-standard hatch patterns that you can use to differentiate the components of objects or represent object materials. The program also comes with 14 hatch patterns that conform to the ISO (International Standards Organization) standards. When you select an ISO pattern, you can specify a pen width, which determines the linewidth in the pattern.

On the Hatch tab of the Hatch and Gradient dialog box, the Type and Pattern area displays the names of all the hatch patterns defined in the *acad.pat* text file. You can add new hatch patterns to the dialog box by adding their definitions to the *acad.pat* file.

Create Associative Hatches

An *associative* hatch is updated when you change the boundary. Hatched areas created with HATCH are associative by default. This setting is stored in the system variable HPASSOC. Hatches created by dragging hatch patterns from tool palettes or DesignCenter™ use the setting in HPASSOC. You can remove hatch associativity at any time or use HATCH to create a nonassociative hatch. When the HPGAPTOL system variable is set to 0 (the default), associativity is automatically removed if editing creates an open boundary.

You can use HATCH to create nonassociative hatches, which are independent of their boundaries.



Assign a Draw Order to a Hatch

You can assign a draw order to a hatch so that it is drawn either behind or in front of the hatch boundary, or behind or in front of all other objects.

When you create a hatch, by default the hatch is drawn behind the hatch boundary. This makes it easier to view and select the hatch boundary. You can change the draw order of the hatch so that the hatch is drawn in front of the hatch boundary instead, or either behind or in front of all other objects. This setting is stored in the HPDRAWORDER system variable. Hatches created by dragging hatch patterns from tool palettes or DesignCenter use the draw order setting in HPDRAWORDER.

Limit Hatch Pattern Density

If you create a very dense hatch, the program may reject the hatch and display a message indicating that the hatch scale is too small or its dash length too short. You can change the maximum number of hatch lines by setting the MaxHatch system registry variable at the Command prompt by entering **(setenv "MaxHatch" "n")** where n is a number between 100 and 10000000 (ten million). The default value for MaxHatch is 10000.

NOTE When changing the value of MaxHatch, you must enter MaxHatch with the capitalization as shown.

Edit Hatch Boundaries

Because there are so many combinations of objects that can be hatched, editing hatched geometry can produce unexpected results. If you create a hatch that you don't want, you can undo it, trim the hatch, or delete the hatch and rehatch the area.

Create Custom Hatch Patterns

You can also define your own hatch pattern using the current linetype with the User Defined Pattern option of the Hatch and Gradient dialog box, or you can create more complex hatch patterns.

See also:

["Modify Hatches and Solid-Filled Areas" on page 787](#)

["Overview of Hatch Pattern Definitions" in the *Customization Guide*](#)

To drag hatch patterns into your drawing

- 1 Click Tools menu ► Palettes ► DesignCenter.

NOTE This procedure describes how to use DesignCenter to drag hatch patterns into your drawing. You can also drag hatch patterns from a tool palette.

- 2 On the Folders tab, click the Search button.
- 3 In the Search dialog box, make the following entries:
 - In Look For, select Hatch Pattern Files.
 - In the In box, select the drive where the program is installed.
 - Select the Search Subfolders option.
 - On the Hatch Pattern Files tab, in Search for the Name, enter * (asterisk).
- 4 Click Search Now.

The default hatch pattern file is *acad.pat* or *acadiso.pat*. The search results may display the same file in different locations.

NOTE For convenient access, you can add the PAT file to Favorites by selecting the file and clicking the Favorites button. A shortcut to the PAT file is displayed in the *Favorites* folder on the Folders tab in DesignCenter.

- 5 In the search results, double-click the file to load the hatch patterns into the content area of DesignCenter.
- 6 (Optional) Right-click a pattern to display a shortcut menu with the following options:
 - **HATCH**. Opens the Hatch and Gradient dialog box.
 - **Copy**. Stores the hatch pattern to the Clipboard.

■ **Create Tool Palette.** Creates a new tool palette with the selected pattern displayed.

- 7 From the content area, drag a hatch pattern onto a closed object in your drawing or onto a tool palette.

NOTE If the hatch pattern scale is too large or small, an error message is displayed. You can adjust the scale for any hatch pattern by double-clicking it to display the Hatch and Gradient dialog box.

Standard toolbar



Command line: ADCENTER

To hatch areas

- 1 Click Draw menu ► Hatch.
- 2 In the Hatch and Gradient dialog box, click Add: Pick points.
- 3 In your drawing, specify a point inside each area that you want hatched, and then press ENTER.
This point is known as the internal point.
- 4 In the Hatch and Gradient dialog box, Hatch tab, in the swatch box, verify that the sample pattern is the pattern you want to use. To change patterns, select another pattern from the Pattern list.
To see how the hatch pattern will look, click the [...] button next to Pattern. Click OK when you finish previewing.
- 5 In the Hatch and Gradient dialog box, make adjustments, if necessary.
You can specify new hatch boundaries by clicking Add Boundaries or Remove Boundaries.
- 6 Under Draw Order, click one of the options.
You can change the draw order of the hatch so that the hatch is drawn either behind or in front of the hatch boundary, or behind or in front of all other objects.
- 7 Click OK.

Draw toolbar



Command line: HATCH

To hatch selected objects

- 1 Click Draw menu ▶ Hatch.
- 2 In the Hatch and Gradient dialog box, click Add: Select objects.
- 3 Specify the object or objects you want to hatch.
The objects need not form a closed boundary. You can also specify any islands that should remain unhatched. Also, you can set the HPGAPTOL system variable to treat a set of objects that almost enclose an area as a closed hatch boundary.
- 4 Under Draw Order, click one of the options.
You can change the draw order of the hatch so that the hatch is drawn either behind or in front of the hatch boundary, or behind or in front of all other objects.
- 5 Click OK.

Draw toolbar



Command line: HATCH

Define Hatch Boundaries

You create a hatch by selecting an object to hatch or fill, or by defining a boundary and then specifying an internal point.

Overview of Hatch Boundaries

You can hatch an enclosed area or hatch within a specified boundary using HATCH. By default, HATCH creates associative hatches that are updated when the boundary is changed.

You create a hatch by selecting an object to hatch or by defining a boundary and then specifying an internal point. A hatch boundary can be any combination of objects, such as lines, arcs, circles, and polylines, that forms an enclosed area.

Enclosed areas within the hatch area are referred to as islands. You can hatch them or leave them unhatched depending on the Islands setting in the Hatch and Gradient dialog box.

If you are hatching a small area in a complex drawing, you can use boundary sets to speed the process.

Objects can be hatched only if they are in a plane parallel to the XY plane of the current UCS.

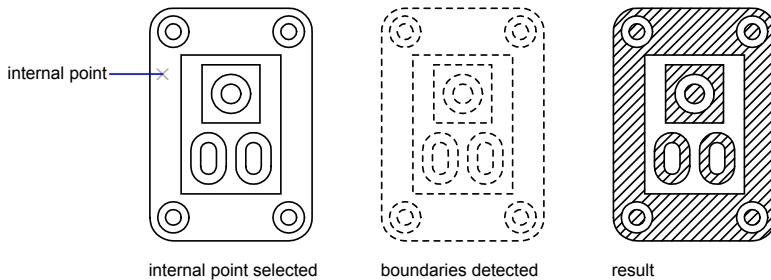
NOTE If you want to hatch an area that is not completely enclosed, you can set a gap tolerance (HPGAPTOL system variable). Any gaps equal to or smaller than the value you specify in the gap tolerance are ignored, and the boundary is treated as closed.

Control the Hatching in Islands

You can determine how *islands*, enclosed areas within the hatch boundary, are hatched using the three hatching styles: Normal, Outer, and Ignore. You can preview these hatching styles in the More Options area of the Hatch and Gradient dialog box.

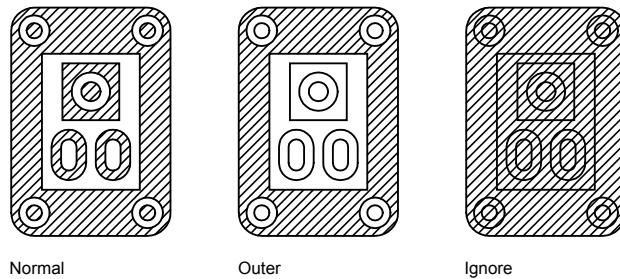
Normal hatching style (the default) hatches inward from the outer boundary. If the hatching process encounters an internal boundary, hatching is turned off until another boundary is encountered.

If you hatch using the Normal hatching style, islands remain unhatched and islands within islands are hatched, as shown below.

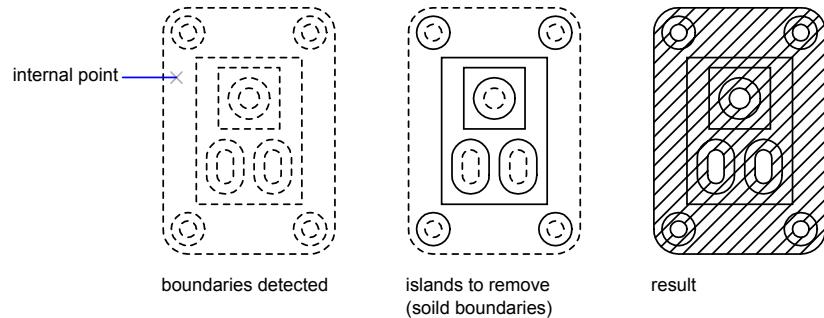


Outer hatching style hatches inward from the outer boundary and stops at the next boundary.

Ignore hatching style hatches the entire enclosed area, ignoring internal boundaries.



You can also remove any islands from the hatch area.



See also:

["Modify Hatches and Solid-Filled Areas" on page 787](#)

To remove islands from the hatch area

- 1 Click Draw menu ▶ Hatch.
- 2 In the Hatch and Gradient dialog box, after adding one or more boundaries, click Remove Boundaries.
- 3 Select the boundaries that you want to remove and press ENTER.
- 4 In the Hatch and Gradient dialog box, click OK to apply the hatch.

Draw toolbar



Command line: HATCH

Define Hatch Boundaries in Large Drawings

You can save time hatching a small area in a complex drawing by defining a set of objects in the drawing to be used in determining the hatch boundary.

By default, HATCH defines the boundary by analyzing all closed objects in the drawing. Analysis of all objects fully or partially visible on the screen as boundaries can be time consuming in a complex drawing. To hatch a small area of a complex drawing, you can define a set of objects in the drawing called a *boundary set*. HATCH does not analyze objects that are not included in the boundary set.

For clarity, first zoom into the area you want to hatch.



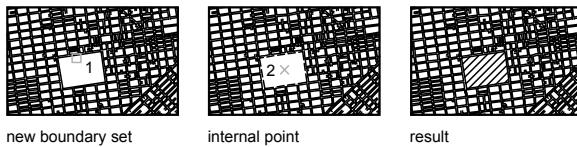
The View Selections option in the Hatch and Gradient dialog box, highlights the objects in the drawing that define the boundary.

To define a boundary set in a complex drawing

- 1 Click Draw menu ► Hatch.
- 2 In the Hatch and Gradient dialog box, More Options, under Boundary Set, click New.
- 3 At the Select Objects prompt, specify opposite corner points for the boundary set and press ENTER.

If you use crossing selection by specifying points from right to left, you select all objects enclosed or crossed.

- 4 In the Hatch and Gradient dialog box, click Add Boundary. If necessary, enter **k** to specify the Pick Internal Point option.
- 5 Specify the internal point.
- 6 Click OK to apply the hatch.



Draw toolbar



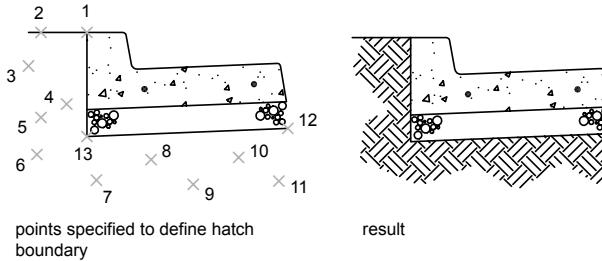
Command line: HATCH

Create Unbounded Hatches

There are several methods you can use to create a hatch that does not display a hatch boundary.

- You can create a hatch with HATCH, and then erase some or all of the boundary objects.
- You can create a hatch with HATCH, making sure that the boundary objects are on a different layer than the hatch. Then turn off or freeze the layer of the boundary objects. This is the only method that maintains hatch associativity.
- You can trim an existing hatch with objects created as trim boundaries. After trimming the hatch, erase the objects.
- You can define a hatch boundary with the Draw option of -HATCH on the command line by specifying boundary points.

For example, you might want to show that a large area of a drawing is filled with a pattern by filling only a small section of that area, as shown in the following illustration.



You can choose whether to retain the polyline boundary after the hatch is created; here, the polyline boundary is not retained.

To define a boundary by specifying points

- 1 At the Command prompt, enter **-hatch**.
- 2 Enter the desired pattern. For example, enter **earth** to select the EARTH pattern.
- 3 Specify the scale and angle for the pattern.
- 4 Enter **w** to specify the Draw option.
- 5 Specify points to define the boundary. Enter **c** to close the polyline boundary, and then press ENTER.
- 6 Enter **n** to discard the polyline boundary once the hatch area has been defined, or enter **y** to create a polyline.

Command line: -HATCH

Choose Hatch Patterns and Solid Fills

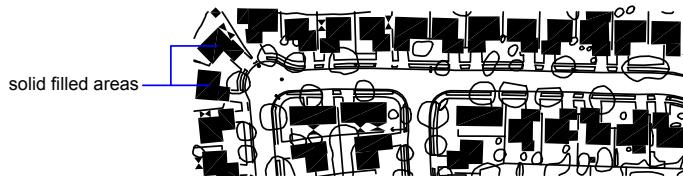
You can use a predefined hatch pattern or solid fill, or you can create your own hatch patterns.

Create Solid-Filled Areas

Solid-filled areas can be created using

- Hatches with a solid hatch pattern (HATCH)
- 2D solids (SOLID)

- Wide polylines or donuts (PLINE, DONUT)



See also:

- "Overview of Hatch Patterns and Fills" on page 770
- "Modify Hatches and Solid-Filled Areas" on page 787
- "Draw Polylines" on page 419
- "Draw Donuts" on page 442

To create a hatch with a solid pattern

- 1 Click Draw menu > Hatch.
- 2 In the Hatch and Gradient dialog box, click Add: Pick points.
- 3 Specify a point in your drawing inside each area that you want hatched. This point is known as the internal point.
- 4 Press ENTER.
- 5 In the Hatch and Gradient dialog box, Hatch tab, under Type, click Predefined.
- 6 Click the [...] button next to Pattern.
- 7 In the Hatch Pattern Palette dialog box, Other Predefined tab, select Solid. Click OK.
- 8 To see how the hatch pattern will look, click Preview.
- 9 When you finish previewing the hatch pattern, right-click or press ENTER to apply the hatch, or press any other button or key to return to the Hatch and Gradient dialog box.
- 10 In the Hatch and Gradient dialog box, make adjustments, if necessary. (You can specify new hatch boundaries by clicking Add Boundaries or Remove Boundaries.)
- 11 Click OK.

Draw toolbar

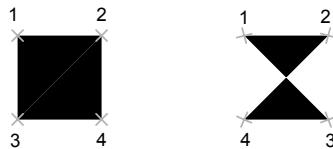


Command line: HATCH

To create a 2D solid object

- 1 At the Command prompt, enter **SOLID**.
- 2 Specify the first point.
- 3 Specify the second point, moving left to right.
- 4 Continue to specify points. Press ENTER when the object is complete.

When you create a quadrilateral solid-filled area, the sequence of the third and fourth points determines its shape. Compare the following illustrations:



Notice that to create the quadrilateral area, both the top and bottom edges are specified from left to right. If you specify the first point on the right and the second point on the left, then the third and fourth points should also be in a right-to-left direction. As you continue to specify pairs of points, be sure to continue this zigzag sequence to ensure the results you expect.

Command line: SOLID

To create a wide polyline

- 1 Click Draw menu ► Polyline.
- 2 Specify the start point of the line segment.
- 3 Enter **w** (Width).
- 4 Enter the starting width of the line segment.

- 5 Specify the ending width of the line segment using one of the following methods:
 - To create a line segment of equal width, press ENTER.
 - To create a tapering line segment, enter a different width.
- 6 Specify the endpoint of the polyline segment.
- 7 Continue specifying segment endpoints as needed.
- 8 Press ENTER to end, or enter **c** to close the polyline.

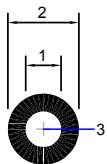
Draw toolbar



Command line: PLINE

To create a donut

- 1 Click Draw menu ▶ Donut.
- 2 Specify the inside diameter (1).
- 3 Specify the outside diameter (2).
- 4 Specify the center of the donut (3).
- 5 Specify the center point for another donut, or press ENTER to complete the command.



Command line: DONUT

Create Gradient-Filled Areas

A gradient fill is a solid hatch fill that gives the blended-color effect of a surface with light on it. You can use gradient fills to suggest a solid form in two-dimensional drawings.

The color in a gradient fill makes a smooth transition from light to dark, or from dark to light, and back. You select a predefined pattern (for example, linear, spherical, or radial sweep) and specify an angle for the pattern. In a two-color gradient fill, the transition is both from light to dark and from the first color to the second.

Gradient fills are applied to objects in the same way solid fills are and can be associated with their boundaries or not. An associated fill is automatically updated when the boundary changes.

You cannot use plot styles to control the plotted color of gradient fills.

Double-click a gradient fill to modify it.

To create a one-color gradient fill

- 1 Click Draw menu ► Hatch.
- 2 In the Hatch and Gradient dialog box, click Add: Pick points or Add: Select objects.
- 3 Specify an internal point or select an object, and then press ENTER.
- 4 In the Hatch and Gradient dialog box, Gradient tab, select One Color.
- 5 If you want to change the color, click the [...] button next to the color.
- 6 In the Select Color dialog box, use the Shade/Tint slider to adjust the color.
 - Move the slider toward Tint to create a color transition that moves toward white.
 - Move the slider toward Shade to create a transition that moves toward black.
- 7 Click a pattern, and set the following options:
 - Select Center to create a symmetrical fill, or clear Center to move the “highlight” up and to the left.
 - Specify an angle for the “highlighted” area.
- 8 To see how the gradient fill will look, click Preview. Press ENTER or right-click to return to the dialog box and make adjustments.
- 9 When you are satisfied, in the Hatch and Gradient dialog box, click OK to create the gradient fill.

Draw toolbar



Command line: HATCH

To create a two-color gradient fill

- 1 Click Draw menu ▶ Hatch.
- 2 In the Hatch and Gradient dialog box, click Add: Pick points or Add: Select objects.
- 3 Specify an internal point or select an object, and then press ENTER.
- 4 In the Hatch and Gradient dialog box, Gradient tab, select Two Color. The second color is the color of the highlighted area in the gradient fill.
- 5 If you want to change either color, click the [...] button next to the color to open the Select Color dialog box.
- 6 Click a pattern, and set the following options:
 - Select Center to create a symmetrical fill, or clear Center to move the “highlight” up and to the left.
 - Specify an angle for the “highlighted” area.
- 7 To see how the gradient fill will look, click Preview. Press ENTER or right-click to return to the dialog box and make adjustments.
- 8 When you are satisfied, in the Hatch and Gradient dialog box, click OK to create the gradient fill.

Draw toolbar



Command line: HATCH

To modify a gradient fill

- Double-click a gradient fill to modify it.

Use Predefined Hatch Patterns

The program supplies a solid fill and more than 50 industry-standard hatch patterns, which can represent materials such as earth, brick, or clay.

Fourteen hatch patterns conform to International Standards Organization (ISO) standards. When you select an ISO pattern, you can specify a pen width, which determines the linewidth in the pattern.

In addition to using the patterns supplied with the program, you can use patterns from an external pattern library. These patterns are listed by name and are displayed in the Hatch Pattern Palette dialog box.

To use a predefined hatch pattern

- 1 Click Draw Menu ► Hatch
- 2 In the Hatch and Gradient dialog box, click Add: Pick points or Add: Select objects.
- 3 Specify an internal point or select an object.
- 4 In the Hatch and Gradient dialog box, Hatch tab, select Predefined in the Type box.
- 5 From the Pattern box, select a pattern.
- 6 Click OK.

Draw toolbar



Command line: HATCH

Create User-Defined Hatch Patterns

In addition to using predefined hatch patterns, you can define a simple hatch pattern based on the current linetype. You define the pattern by changing the angle and spacing of the hatch lines.

To create a user-defined hatch pattern

- 1 Specify the linetype for the user-defined hatch pattern by making that linetype current.
- 2 Click Draw Menu ► Hatch.
- 3 In the Hatch and Gradient dialog box, click Add: Pick points or Add: Select objects.
- 4 Specify an internal point or select an object.

- 5 In the Hatch and Gradient dialog box, Hatch tab, select User-Defined in the Type box.
- 6 Specify the angle and spacing of the hatch pattern.
- 7 To use intersecting lines in the pattern, select Double.
- 8 Click OK.

Draw toolbar



Command line: HATCH

Modify Hatches and Solid-Filled Areas

You can modify both the pattern fill and the boundaries of hatches. You can also modify solid-filled areas, but the method you use depends on whether the solid-filled area is a solid-filled hatch, a 2D solid, or a wide polyline or donut. You can also modify the draw order of your hatch.

Control Hatch Pattern Density

Hatching can produce a very large number of line and point objects. Although stored as hatch objects, these line and point objects do use disk space and take time to generate. If you use a relatively small scale factor when hatching an area, the hatch could require millions of line and point objects, thus taking a very long time to complete and possibly exhausting the available resources. You can avoid this problem by imposing a limit on the number of objects created by a single HATCH command. If the approximate number of objects needed for a particular hatch (considering the boundary extents, pattern, and scale) exceeds the limit, HATCH displays a message indicating that the hatch scale is too small or that its dash length is too short, and the hatch request is rejected. If this occurs, carefully examine your hatch settings. The scale factor may be unreasonable and may need to be adjusted.

The hatch object limit is set by the MaxHatch environment setting, which is stored in the system registry. Its default value is 10000. You can change this limit by setting the MaxHatch system registry variable using (**setenv "MaxHatch" "n"**) where *n* is a number between 100 and 10000000 (ten million).

Change the Hatch Properties of an Existing Hatch

You can modify hatch-specific properties, such as pattern, scale, and angle, of an existing hatch. You can use the following:

- Hatch Edit dialog box (recommended)
- Properties palette

You can also copy properties from one hatch to another. With the Inherit Properties button in the Hatch Edit dialog box, you can copy all hatch-specific properties, including the hatch origin, from one hatch to another. Use the Match Properties dialog box to copy general properties and hatch-specific properties (with the exception of the hatch origin), from one hatch to another.

You can use EXPLODE to disassemble a hatch into its component objects.

Modify a Hatch Boundary

Hatch boundaries can be copied, moved, stretched, trimmed, and so on. You can also use grips to stretch, move, rotate, scale, and mirror hatch boundaries and their associated hatches just as you do other objects. If the editing you do maintains a closed boundary, an associative hatch is updated automatically. If the editing produces an open boundary, the hatch loses any associativity with the boundary and remains unchanged. Associativity may also be lost during editing of a hatch boundary if the hatch pattern file is not available at the time of editing. If you trim a hatch and the hatch pattern (PAT) file is no longer available, the hatch will disappear.

Hatch associativity depends on whether you choose Associative in the Hatch and Gradient (HATCH) and Hatch Edit (HATCHEDIT) dialog boxes.

Nonassociative hatches are not updated when their original boundary is changed.

You can remove hatch associativity at any time, but once it is removed for an existing hatch, it cannot be reestablished. The hatch must be re-created to restore associativity or a new hatch boundary must be created and associated with the hatch.

To create a boundary around a nonassociative or unbounded hatch, in the Hatch and Gradient dialog box, use the Recreate Boundary option. With this option you can also specify that the new boundary is associated with the hatch.

Modify Solid-Filled Areas

Solid-filled areas can be represented by

- Hatches (with a solid hatch pattern)

- 2D solids
- Gradient fills
- Wide polylines or donuts

You modify each of these solid-filled objects just as you would any other hatch, 2D solid, wide polyline, or donut. In addition to PROPERTIES, you can use HATCHEDIT for solid-filled hatches and gradient fills, grip editing for 2D solids, and PEDIT for wide polylines and donuts.

Modify the Draw Order of a Hatch

When you edit a hatch, you can change its draw order, so that it is displayed behind the hatch boundary, in front of the hatch boundary, behind all other objects, or in front of all other objects.

To change the angle of a hatch

- 1 Select the hatch pattern.
- 2 Right-click the hatch. Click Properties.
- 3 In the Properties palette, enter the new value for Angle.

Standard toolbar



Command line: PROPERTIES

To set the limit of objects in a hatch pattern

- At the Command prompt, enter (**setenv “MaxHatch” “n”**)

where n is a number between 100 and 10000000 (ten million). The higher the value, the more dense the hatch pattern.

NOTE MaxHatch is case-sensitive.

To modify a gradient fill

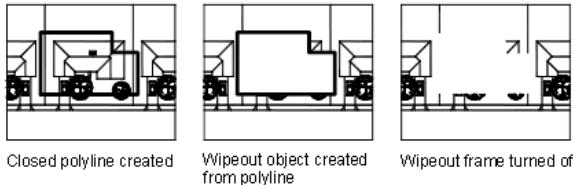
- 1 Double-click the gradient fill you want to modify.
- 2 In the Hatch Edit dialog box, Gradient tab, make any changes.

- To see how the gradient fill will look, click Preview. Press ENTER or right-click to return to the dialog box and make adjustments.
- When you are satisfied, in the Hatch and Gradient dialog box, click OK to create the gradient fill.

Create a Blank Area to Cover Objects

A wipeout object is a polygonal area that masks underlying objects with the current background color. This area is bounded by the wipeout frame, which you can turn on for editing and turn off for plotting.

You can create a wipeout object by specifying a polygonal area with a series of points, or you can convert a closed polyline into a wipeout object.



Requirements and Limitations

If a polyline is used to create a wipeout object, the polyline must be closed, contain line segments only, and have zero width.

You can create wipeout objects on a layout in paper space to mask objects in model space; however, on the Plot Settings tab of the Plot dialog box, the Plot Paper Space Last option must be cleared before you plot to ensure that the wipeout object is plotted correctly.

Because a wipeout object is similar to a raster image, it has the same requirements for plotting: you need a raster-capable plotter with either an ADI 4.3 raster-capable driver or the system printer driver.

See also:

[Control How Overlapping Objects Are Displayed](#)

To cover existing objects with a blank area

- Click Draw menu ► Wipeout.
- Specify points in a sequence that defines the perimeter of the area to be masked.

3 Press ENTER to end.

Command line: WIPEOUT

To turn all wipeout frames on or off

- 1** Click Draw menu ► Wipeout.
- 2** On the command line, enter **f** (Frames).
- 3** Enter **on** or **off** and press ENTER.

Command line: WIPEOUT

Notes and Labels

26

You can create and modify several types of text, including text with leaders. You can control most text style settings by defining text styles.

In this chapter

- Overview of Notes and Labels
- Create Text
- Use Fields in Text
- Work with Text Styles
- Change Text
- Check Spelling
- Use an Alternate Text Editor

Overview of Notes and Labels

You can create text in various ways. For short, simple entries, use single-line text. For longer entries with internal formatting, use multiline text (also called *mtext*). You can also create multiline text with leaders.

Although all entered text uses the current text style, which establishes the default font and format settings, you can use several methods to customize the text appearance. There are several tools that can change text scale and justification, find and replace text, and check for spelling errors.

Text that is included in a dimension or tolerance is created using the dimensioning commands.

Create Text

You can create text using several methods, depending on your needs.

See also:

["Use Fields in Text" on page 823](#)

Overview of Creating Text and Leaders

The text you add to your drawings conveys a variety of information. It may be a complex specification, title block information, a label, or even part of the drawing.

Single-Line Text

For short entries that do not require multiple fonts or lines, create single-line text. Single-line text is most convenient for labels.

Multiline Text

For long, complex entries, create multiline, or paragraph text. Multiline text consists of any number of text lines or paragraphs that fit within a width you specify; it can extend vertically to an indefinite length.

Regardless of the number of lines, each set of paragraphs created in a single editing session forms a single object, which you can move, rotate, erase, copy, mirror, or scale.

There are more editing options for multiline text than there are for single-line text. For example, you can apply underlining, fonts, color, and text height changes to individual characters, words, or phrases within a paragraph.

Leader Objects

A leader object is a line or a spline with an arrowhead at one end and a multiline text object at the other. The leader object is associated with the multiline text object, so when the text object is moved, rotated, or scaled, the leader object is updated accordingly. Similarly, when associative dimensioning is turned on, and object snaps are used to locate the leader arrowhead, a leader object is also associated with any objects to which the arrowhead is attached.

You can copy text used elsewhere in a drawing and append a leader to it.

NOTE The leader object should not be confused with the leader line that is automatically generated as part of a dimension line.

Create Single-Line Text

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** on the command line to enter the text on the command line.

You can insert a field in single-line text. A field is text that is set up to display data that might change. When the field is updated, the latest value of the field is displayed.

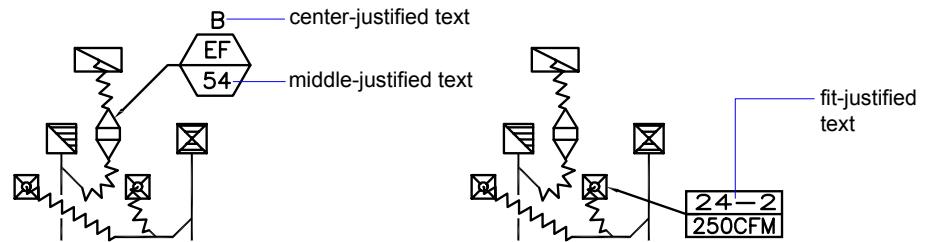
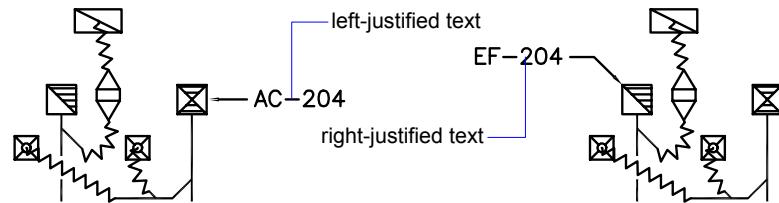
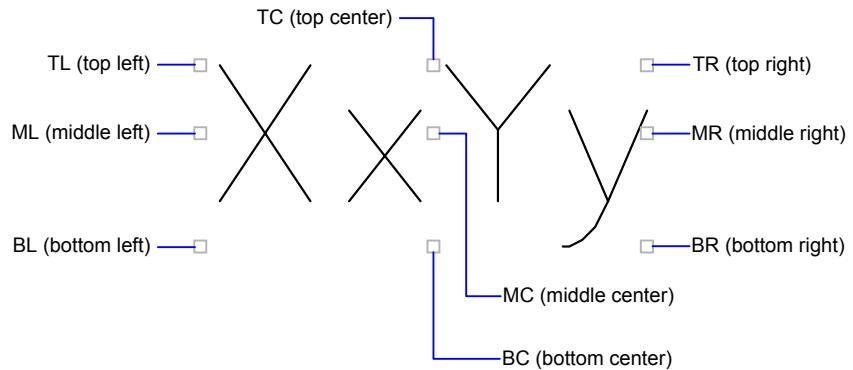
The text styles used for single-line text are the same as those used for multiline text. When you create text, you assign an existing style by entering its name at the Style prompt. If you need to apply formatting to individual words and characters, use multiline text instead of single-line text.

You can also compress single-line text to fit between points that you specify. This option stretches or squeezes the text to fill the designated space.

The DTEXTED system variable specifies the user interface displayed for editing single-line text.

Align Single-Line Text

As you create text, you can align it. That is, you can justify it with one of the alignment options shown in the following illustrations. Left alignment is the default. To left-align text, do not enter an option at the Justify prompt.



See also:

["Use Fields in Text" on page 823](#)

To create single-line text

- 1 Click Draw menu > Text > Single Line Text.

2 Specify the insertion point for the first character. If you press ENTER, the program locates the new text immediately below the last text object you created, if any.

3 Specify the height of the text. This prompt is displayed only if text height is set to 0 in the current text style.

A rubber-band line is attached from the text insertion point to the cursor. Click to set the height of the text to the length of the rubber-band line.

4 Specify a text rotation angle.

You can enter an angle value or use your pointing device.

5 Enter the text. At the end of each line, press ENTER. Enter more text as needed.

NOTE Text that would otherwise be difficult to read (if it is very small, very large, or is rotated) is displayed at a legible size and is oriented horizontally so that you can easily read and edit it.

If you specify another point during this command, the cursor moves to that point, and you can continue typing. Every time you press ENTER or specify a point, a new text object is created.

6 Press ENTER on a blank line to end the command.

Command line: TEXT

To specify a text style when you create single-line text

1 Click Draw menu ▶ Text ▶ Single Line Text.

2 Enter **s** (Style).

3 At the Style Name prompt, enter an existing text style name.

If you first want to see a list of text styles, enter **?** and press ENTER twice.

4 Continue creating text.

Command line: TEXT

To align single-line text as you create it

1 Click Draw menu ▶ Text ▶ Single Line Text.

2 Enter **j** (Justify).

3 Enter an alignment option. For example, enter **br** to align text at its bottom-right corner.

4 Continue creating text.

Command line: TEXT

Create Multiline Text

A multiline text (mtext) object includes one or more paragraphs of text that can be manipulated as a single object.

Overview of Multiline Text

You can create one or more paragraphs of multiline text (mtext) in the In-Place Text Editor (or an alternative text editor) or use prompts on the command line. You can also insert text from a file saved in ASCII or RTF format.

Before entering or importing text, you specify opposite corners of a text bounding box that defines the width of the paragraphs in the multiline text object. The length of the multiline text object depends on the amount of text, not the length of the bounding box. You can use grips to move or rotate a multiline text object.

The In-Place Text Editor displays the bounding box with a ruler at the top and the Text Formatting toolbar. The editor is transparent so that, as you create text, you can see whether the text overlaps other objects. To turn off transparency while you work, check Opaque Background on the Options menu. You can also make the background of the finished multiline text object opaque and set its color.

You can set tabs and indent text to control the appearance of paragraphs in the multiline text object.

You can also insert fields in multiline text. A field is text that is set up to display data that might change. When the field is updated, the latest value of the field is displayed.

Text Style

Most characteristics of the text are controlled by the text style, which sets the default font and other options, such as line spacing, justification, and color. You can use the current text style or select a new one. The STANDARD text style is the default.

Within the multiline text object, you can override the current text style by applying formatting such as underlining, boldface, and different fonts to individual characters. You can also create stacked text, such as fractions or geometric tolerances and insert special characters, including Unicode characters, for TrueType fonts.

NOTE Not all SHX and TrueType text fonts support Unicode characters.

Text Properties

In the Properties palette, you can view and change the object properties of a multiline text object, including properties that apply specifically to text.

- Justification determines where text is inserted with respect to the bounding box and sets the direction of text flow as text is entered.
- Line space options control the amount of space between lines of text.
- Width defines the width of the bounding box and therefore controls where the text wraps to a new line.
- Background inserts an opaque background so that objects under the text are masked.

To create multiline text

- 1 Click Draw menu ► Text ► Multiline Text.
- 2 Specify opposite corners of a bounding box to define the width of the multiline text object.
The In-Place Text Editor is displayed.
- 3 To indent the first line of each paragraph, drag the first-line indent slider on the ruler. To indent the other lines of each paragraph, drag the paragraph slider.
- 4 To set tabs, click the ruler where you want a tab stop.
- 5 If you want to use a text style other than the default, click the arrow next to the Text Style control on the toolbar and then select a style.
- 6 Enter text.

NOTE Text that would otherwise be difficult to read (if it is very small, very large, or is rotated) is displayed at a legible size and is oriented horizontally so that you can easily read and edit it.

- 7 To override the current text style, select text as follows:
 - To select one or more letters, click and drag the pointing device over the characters.
 - To select a word, double-click the word.
 - To select a paragraph, triple-click the paragraph.

- 8** On the toolbar, make format changes as follows:
 - To change the font of the selected text, select a font from the list.
 - To change the height of the selected text, enter a new value in the Height box.
 - To format text in a TrueType font with boldface or italics, or to create underlined or overlined text for any font, click the corresponding button on the toolbar. SHX fonts do *not* support boldface or italics.
 - To apply color to selected text, choose a color from the Color list. Click Other to display the Select Color dialog box.
- 9** To save your changes and exit the editor, use one of the following methods:
 - Click OK on the toolbar.
 - Click in the drawing outside the editor.
 - Press CTRL+ENTER.

Draw toolbar



Command line: MTEXT

To insert symbols or special characters in multiline text

- 1** Double-click a multiline text object to open the In-Place Text Editor.
- 2** On the expanded toolbar, click Symbol.
- 3** Click one of the options on the menu, or click Other to display the Character Map dialog box.

To access the Character Map dialog box, you must have *charmap.exe* installed. See Microsoft® Windows® Help for information about adding programs to your system.
- 4** In the Character Map dialog box, select a font.
- 5** Select a character, and use one of the following methods:
 - To insert a single character, drag the selected character into the editor.
 - To insert multiple characters, click Select to add each character to the Characters to Copy box. When you have all the characters you want, click Copy. Right-click in the editor. Click Paste.

6 To save your changes and exit the editor, use one of the following methods:

- Click OK on the toolbar.
- Click in the drawing outside the editor.
- Press CTRL+ENTER.

Draw toolbar



Command line: MTEXT

To add an opaque background or fill to a multiline text object

1 Double-click a multiline text object to open the In-Place Text Editor.

2 In the editor, right-click. Click Background Mask.

3 In the Background Mask dialog box, select Use Background Mask.

4 Enter a value for Border Offset Factor.

The value is based on the text height. A factor of 1.0 exactly fits the multiline text object. A factor of 1.5 (the default) extends the background by 0.5 times the text height.

5 Under Fill Color, do one of the following:

- Select the Use Drawing Background Color option.
- Select a color for the background, or click Select Color to open the "Select Color Dialog Box".

6 Click OK to return to the editor.

7 To save your changes and exit the editor, use one of the following methods:

- Click OK on the toolbar.
- Click in the drawing outside the editor.
- Press CTRL+ENTER.

The opaque background is applied when you exit the editor.

Draw toolbar



Command line: MTEXT

To change the Windows font smoothing setting to improve visibility of text in the Multiline Text editor

- 1 On the Windows desktop, right-click. Click Properties.
- 2 In the Display Properties dialog box, click the Appearance tab.
- 3 Click Effects.
- 4 In the Effects dialog box, click the Use the Following Method to Smooth Edges of Screen Fonts to clear the setting.
- 5 Click OK to exit the Effects dialog box.
- 6 Click OK to exit the Display Properties dialog box.

Justify Multiline Text

Justification controls both text alignment and text flow relative to the text insertion point. Text is left-justified and right-justified with respect to the boundary rectangle that defines the text width. Text flows from the insertion point, which can be at the middle, the top, or the bottom of the resulting text object.

There are nine justification settings for multiline text.

If a single word is longer than the width of the paragraph, the word will extend beyond the paragraph boundary.

<p><i>Computer security is not a new subject but it has gained more attention recently because of the increased use of computers in business and government. The importance of computer security has increased as more and more sensitive information has been stored in computers. The importance of the function is to protect the system from unauthorized access and to ensure that the data is accurate and reliable.</i></p>	<p><i>The importance of the function is to protect the system from unauthorized access and to ensure that the data is accurate and reliable. The function is implemented by using various security measures such as firewalls, encryption, and access controls. The function is also used to detect and prevent attacks on the system.</i></p>	<p><i>Computer security is not a new subject but it has gained more attention recently because of the increased use of computers in business and government. The importance of computer security has increased as more and more sensitive information has been stored in computers. The importance of the function is to protect the system from unauthorized access and to ensure that the data is accurate and reliable.</i></p>
<p>top left: left-justified, spills down</p>	<p>top center: center-justified, spills down</p>	<p>top right: right-justified, spills down</p>
<p><i>Computer security is not a new subject but it has gained more attention recently because of the increased use of computers in business and government. The importance of computer security has increased as more and more sensitive information has been stored in computers. The importance of the function is to protect the system from unauthorized access and to ensure that the data is accurate and reliable.</i></p>	<p><i>The importance of the function is to protect the system from unauthorized access and to ensure that the data is accurate and reliable. The function is implemented by using various security measures such as firewalls, encryption, and access controls. The function is also used to detect and prevent attacks on the system.</i></p>	<p><i>Computer security is not a new subject but it has gained more attention recently because of the increased use of computers in business and government. The importance of computer security has increased as more and more sensitive information has been stored in computers. The importance of the function is to protect the system from unauthorized access and to ensure that the data is accurate and reliable.</i></p>
<p>middle left: left-justified, spills up and down</p>	<p>middle center: center-justified, spills up and down</p>	<p>middle right: right-justified, spills up and down</p>
<p><i>Computer security is not a new subject but it has gained more attention recently because of the increased use of computers in business and government. The importance of computer security has increased as more and more sensitive information has been stored in computers. The importance of the function is to protect the system from unauthorized access and to ensure that the data is accurate and reliable.</i></p>	<p><i>The importance of the function is to protect the system from unauthorized access and to ensure that the data is accurate and reliable. The function is implemented by using various security measures such as firewalls, encryption, and access controls. The function is also used to detect and prevent attacks on the system.</i></p>	<p><i>Computer security is not a new subject but it has gained more attention recently because of the increased use of computers in business and government. The importance of computer security has increased as more and more sensitive information has been stored in computers. The importance of the function is to protect the system from unauthorized access and to ensure that the data is accurate and reliable.</i></p>
<p>bottom left: left-justified, spills up</p>	<p>bottom center: center-justified, spills up</p>	<p>bottom right: right-justified, spills up</p>

To justify multiline text

- 1 Click Tools menu ► Properties.
 - 2 Select the multiline text object.
 - 3 On the Properties palette, select one of the Justification options.
 - 4 Click outside the Properties palette.

Command line: PROPERTIES

Format Characters Within Multiline Text

As you create multiline text, you can override the text style and apply different formatting to individual words and characters. The format changes affect only the text you select; the current text style is not changed.

You can specify a different font and text height and apply boldface, italics, underlining, overlining, and color. You can also set an obliquing angle, change the space between characters, and make characters wider or narrower. The Remove Formatting option on the menu of options resets the character attributes of selected text to the current text style and also resets the text color to the color of the mtext object.

The text height setting specifies the height of capitalized text. For more information about how height is calculated, see MTEXT.

See also:

["Use an Alternate Text Editor" on page 850](#)

To format characters in multiline text

- 1** Double-click a multiline text object to open the In-Place Text Editor.
- 2** Select the text you want to format:
 - To select one or more letters, click and drag the pointing device over the characters.
 - To select a word, double-click the word.
 - To select a paragraph, triple-click the paragraph.
- 3** On the toolbar, make format changes as follows:
 - To change the font of the selected text, select a font from the list.
 - To change the height of the selected text, enter a new value in the Height box.
 - To format text in a TrueType font with boldface or italics, or to create underlined or overlined text for any font, click the corresponding button on the toolbar. SHX fonts do *not* support boldface or italics.
 - To apply color to selected text, select a color from the Color list. Click Other to display the "Select Color Dialog Box".
 - To set an obliquing angle for the text, enter a value between -85 and 85. A positive value slants text to the right. A negative value slants text to the left.
 - To change letter spacing in the selected text, enter a new value.
 - To change the width of characters in the selected text, enter a new value.

4 To save your changes and exit the editor, use one of the following methods:

- Click OK on the toolbar.
- Click in the drawing outside the editor.
- Press CTRL+ENTER.

Draw toolbar



Command line: MTEXT

Create Lists in Multiline Text

Lines of multiline text can be formatted as a list. When you add or delete an item, or move an item up or down a level, the list numbering automatically adjusts. You can remove and reapply list formatting with the same method as used in most text editors.

Use Automatic List Formatting

By default, list formatting is applied to all text that looks like a list. Text that meets all the following criteria is considered to be a list:

- The line begins with one or more letters or numbers or a symbol.
- The letters or numbers is followed by punctuation.
- A space after the punctuation is created by pressing TAB.
- The text following the space is ended by ENTER or SHIFT+ENTER.

NOTE If you do not want list formatting applied to all text that fits the criteria, clear the Allow Bullets and Lists option. (Right-click in the In-Place Text Editor, click Bullets and Lists, and clear Allow Bullets and Lists.) When Allow Bullets and Lists is not checked, you cannot create new formatted lists in the multiline text object.

To create a list, use one of the following methods:

- Apply list formatting to new or selected text.
- Use Auto-list (on by default) and type the elements of a list.

- With Auto-list off, type the elements of a list and close and reopen the editor to convert the text to a list.

Apply List Formatting

When you apply list formatting, you can specify bullets, uppercase or lowercase letters, or numbers. Default settings are used for the type of list you choose. Letters or numbers are followed by a period. Nested lists use a double bullet, letter, or number. Items are indented based on the tab stops on the ruler in the In-Place Text Editor.

Use Auto-list to Type a List

When Auto-list is on, you can create a list as you type. You can use letters, numbers, or symbols. For example, you can type 421., press TAB, and enter some text. When you press ENTER, the next line starts with 422, a period, and a TAB space.

The following characters can be used as punctuation after the number or letter when you type a list but cannot be used as bullets:

Character	Description
.	Period
:	Colon
)	Close parenthesis
>	Close angle bracket
]	Close square bracket
}	Close curly bracket

Paste a List from Another Document

If you copy a nested bulleted list (a list within a list) from Microsoft Word and paste the list into multiline text, the bullets that are displayed as empty circles cannot be formatted like other bullets in multiline text. This is because Word uses the letter *o* instead of a bullet for nested bulleted lists. You can remove formatting from the nested list and reapply to change the bullets to double bullets.

To format multiline text as a list

- Click Draw menu ► Text ► Multiline Text.

- 2 Specify opposite corners of a bounding box to define the width of the multiline text object.
- 3 To expand the Text Formatting toolbar, click Options button ➤ Show Options.
- 4 If you are converting multiline text to a list, select the paragraphs.

NOTE List formatting is only available when the Allow Bullets and Lists option is checked (the default).

- 5 On the expanded toolbar, click Numbering, Bullets, or Uppercase Letters.
 - **Numbering.** Uses numbers with periods for the items in a list.
 - **Bullets.** Uses a bullet or other character for the items in a list.
 - **Uppercase Letters.** Uses uppercase letters with periods for the items in a list. If the list has more items than the alphabet has letters, the sequence continues by using double letters. To use lowercase letters, right-click in the editor. Click Bullets and Lists ➤ Lettered ➤ Lowercase.
- 6 If you are creating new list items, enter the text.
- 7 To end the list, press ENTER to move to a new line. Click the button that you clicked to start the list.
- 8 To save your changes and exit the editor, use one of the following methods:
 - Click OK on the toolbar.
 - Click in the drawing outside the editor.
 - Press CTRL+ENTER.

Draw toolbar



Command line: MTEXT

To remove list formatting from multiline text

- 1 Double-click a multiline text object to open the In-Place Text Editor.
- 2 Select the list items.

- 3 To expand the Text Formatting toolbar, click Options button ► Show Options.
- 4 On the expanded toolbar, click the active list button to make it inactive: Numbering, Bullets, or Uppercase Letters.

NOTE If the list uses lowercase letters, click Uppercase Letters to convert the list to uppercase. Then click Uppercase Letters to make it inactive.

- 5 To save your changes and exit the editor, use one of the following methods:
 - Click OK on the toolbar.
 - Click in the drawing outside the editor.
 - Press CTRL+ENTER.

Draw toolbar



Command line: MTEXT

To create a lettered or numbered list in multiline text as you type

- 1 Click Draw menu ► Text ► Multiline Text.
- 2 Specify opposite corners of a bounding box to define the width of the multiline text object.
- 3 To expand the Text Formatting toolbar, click Options button ► Show Options.
- 4 Click Bullets and Lists. Verify that Allow Auto-list and Allow Bullets and Lists are checked.
- 5 Enter a letter or a number and a period (or other punctuation).
The following characters can be used as punctuation after letters and numbers: period (.), colon (:), close parenthesis ()) , close angle bracket (>), close square bracket ([]), and close curly bracket {}.
- 6 Press TAB.
- 7 Enter the text of the list item. Press ENTER to move to the next item, or press SHIFT+ENTER to add a plain paragraph before the next item.
The item is automatically lettered or numbered in sequence.
- 8 Press ENTER twice to end the list.

- 9** To save your changes and exit the editor, use one of the following methods:
- Click OK on the toolbar.
 - Click in the drawing outside the editor.
 - Press CTRL+ENTER.

Draw toolbar



Command line: MTEXT

To create a bulleted list in multiline text as you type

- 1** Click Draw menu ▶ Text ▶ Multiline Text.
- 2** Specify opposite corners of a bounding box to define the width of the multiline text object.
- 3** To expand the Text Formatting toolbar, click Options button ▶ Show Options.
- 4** Click Bullets and Lists. Verify that Allow Auto-list and Allow Bullets and Lists are checked.
- 5** Start a line of text by entering \U+2022 (the Unicode string for a bullet) or by selecting a bullet character or another symbol.

The following characters cannot be used as bullets: period (.), colon (:), close parenthesis ()) , close angle bracket (>), close square bracket () , and close curly bracket {} .
- 6** Press TAB.
- 7** Enter the text of the list item. Press ENTER to move to the next item, or press SHIFT+ENTER to add a plain paragraph before the next item.

The bullet character is automatically added to the next item.
- 8** Press ENTER twice to end the list.
- 9** To save your changes and exit the editor, use one of the following methods:

 - Click OK on the toolbar.
 - Click in the drawing outside the editor.
 - Press CTRL+ENTER.

Draw toolbar



Command line: MTEXT

To move a list item in multiline text down a level

- 1 Double-click a multiline text object to open the In-Place Text Editor.
- 2 Place the cursor at the beginning of a list item and press TAB.
The item moves down one level and begins a nested list.
- 3 Press ENTER to start the next item at the same level, or press SHIFT+TAB to move the item up a level.
- 4 To save your changes and exit the editor, use one of the following methods:
 - Click OK on the toolbar.
 - Click in the drawing outside the editor.
 - Press CTRL+ENTER.

Draw toolbar



Command line: MTEXT

To separate an existing list

- 1 Double-click a multiline text object to open the In-Place Text Editor.
- 2 Select a sequence of list items or place the cursor at the beginning of the item where you want to start the new list.
- 3 Right-click in the editor. Click Bullets and Lists ► Restart.
The selected items are renumbered as a separate sequence. If you select items in the middle of a list, unselected items below the selected items also become part of the new list.
- 4 To continue the original list below the new list, select the first item below the new list.
- 5 Right-click in the editor. Click Bullets and Lists ► Continue.

The selected item and the items following it are renumbered to continue the previous list.

- 6 To save your changes and exit the editor, use one of the following methods:
 - Click OK on the toolbar.
 - Click in the drawing outside the editor.
 - Press CTRL+ENTER.

Draw toolbar



Command line: MTEXT

To convert the lists in a multiline text object to plain text

- 1 Double-click a multiline text object to open the In-Place Text Editor.
- 2 Right-click in the editor. Click Bullets and Lists. Remove the check mark next to Allow Bullets and Lists.

The plain text lists retain their bullets, numbers, or letters. If you add an item to the list, the number or letter sequence does not change.

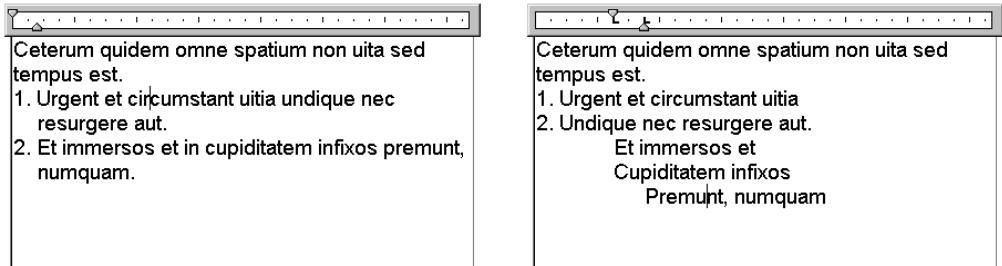
Indent Multiline Text and Use Tabs

You can control how paragraphs are indented in a multiline text (mtext) object. The ruler in the In-Place Text Editor shows the settings for the current paragraph.

Tabs and indents that you set before you start to enter text apply to the whole multiline text object. To apply different tabs and indents to individual paragraphs, click in a single paragraph or select multiple paragraphs and then change the settings.

Sliders on the ruler show indentation relative to the left side of the bounding box. The top slider indents the first line of the paragraph, and the bottom slider indents the other lines of the paragraph.

The long tick marks on the ruler show the default tab stops. If you click the ruler to set your own tabs, the ruler displays a small, L-shaped marker at each custom tab stop. You can delete a custom tab stop by dragging the marker off the ruler.



To create paragraphs with hanging indentation

- 1 Double-click a multiline text object to open the In-Place Text Editor.
- 2 Select the paragraphs you want to indent.
- 3 On the ruler, slide the top indent marker to where you want the first line of the paragraphs to start.

NOTE The ruler displays the tabs and indents that are set for the selected paragraphs or, if no text is selected, the paragraph where the cursor is located. The default tab stops are the long tick marks on the ruler. To set a custom tab stop, click the ruler where you want the tab stop.

- 4 Slide the bottom indent marker to where you want the other lines of the paragraphs to start.

This step indents turnover lines in paragraphs that are more than one line long.

- 5 To change the indentation, select the paragraphs you want to change, click the ruler to set new tab stops, if needed, and move the indent markers.
- 6 To save your changes and exit the editor, use one of the following methods:
 - Click OK on the toolbar.
 - Click in the drawing outside the editor.
 - Press CTRL+ENTER.

Draw toolbar



Command line: MTEXT

Specify the Line Spacing Within Multiline Text

Line spacing for multiline text is the distance between the baseline (bottom) of one line of text and the baseline of the next line of text. The line space factor applies to the entire multiline text object, not to selected lines.

You can set the spacing increment to a multiple of single line spacing, or as an absolute distance. Single spacing is 1.66 times the height of the text characters.

The default line space style, At Least, automatically increases line spacing to accommodate characters that are too large to fit the line spacing you set for the multiline text object. Use the other line space style, Exactly, to line up text in tables.

To ensure that line spacing is identical in multiple multiline text objects, use Exactly and set the Line Space Factor to the same value in each multiline text object.

NOTE Using Exactly can cause text in lines located above or below lines with large font characters to overlap the larger characters.

To change the line spacing of multiline text

- 1 Click Tools menu ▶ Properties.
The Properties palette is displayed.
- 2 Select the multiline text object you want to edit.
- 3 In the Properties palette, for Line Space Style, select one of the following:
 - **At Least.** Adjusts lines of text automatically based on the height of the largest character in the line. More space is added between lines of text with taller characters. This is the default setting.
 - **Exactly.** Forces the line spacing to be the same size for all lines of text regardless of format differences such as font or text height.
- 4 Change the line spacing by entering a new value for either of the following options. The two line spacing options provide different ways to set the same thing:
 - **Line Space Factor.** Sets the line spacing to a multiple of single-line spacing. Single spacing is 1.66 times the height of the text characters.

- **Line Space Distance.** Sets the line spacing to an absolute value measured in drawing units. Valid values must be between 0.0833 and 1.3333.

NOTE After you exit the Properties palette, the value of the *other* line spacing option is updated to correspond with the line spacing value that you entered.

Standard toolbar



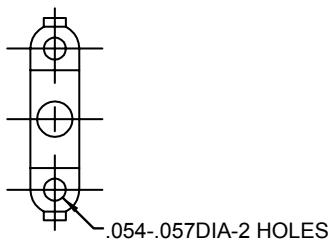
Command line: PROPERTIES

Create Stacked Characters Within Multiline Text

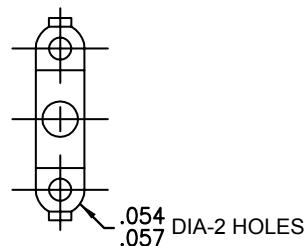
Characters representing a fractional measurement or tolerance can be formatted to conform to several standards.

Stacked text is text or fractions that indicate tolerances or measurements. You use special characters to indicate where selected text should be stacked.

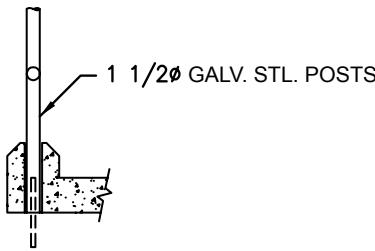
- Slash (/) stacks text vertically, separated by a horizontal line.
- Pound sign (#) stacks text diagonally, separated by a diagonal line.
- Carat (^) creates a tolerance stack, which is not separated by a line.



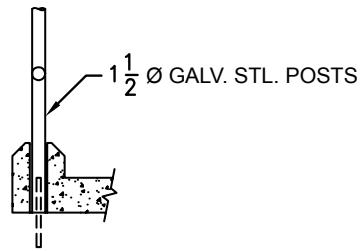
plain text



tolerance stack



diagonal fraction



vertical fraction

You can set the program to automatically stack numeric characters entered before and after the slash, pound sign, or carat. For example, if you enter **1#3** followed by a nonnumeric character or space, the AutoStack Properties dialog box is displayed. You can change the settings to indicate your formatting preferences.

AutoStack stacks only numeric characters immediately before and after the slash, pound sign, and carat. To stack nonnumeric characters or text that includes spaces, select the text and click the Stack button on the Text Formatting toolbar.

See also:

["Use an Alternate Text Editor" on page 850](#)

To create stacked text

- 1 Click Draw menu ▶ Text ▶ Multiline Text.
- 2 Specify opposite corners of a bounding box to define the width of the multiline text object.

- 3** In the In-Place Text Editor, set text style and other multiline text properties as needed.
- 4** Enter the text you want to stack separated by one of the following characters:
 - Slash (/) stacks text vertically, separated by a horizontal line.
 - Pound sign (#) stacks text diagonally, separated by a diagonal line.
 - Carat (^) creates a tolerance stack, which is not separated by a line.If you enter numbers separated by stack characters and then enter a nonnumeric character or press SPACEBAR, the AutoStack Properties dialog box is displayed.
- 5** In the AutoStack Properties dialog box, you can choose to automatically stack numbers (not nonnumeric text) and to remove leading blanks. You can also specify whether the slash character creates a diagonal fraction or creates a vertical fraction. If you do not want to use AutoStack, click Cancel to exit the dialog box.
- 6** Select the text that you want to stack, and click the Stack button on the toolbar.
- 7** To save your changes and exit the editor, use one of the following methods:
 - Click OK on the toolbar.
 - Click in the drawing outside the editor.
 - Press CTRL+ENTER.

Draw toolbar



Command line: MTEXT

To change stack properties

- 1** Double-click the multiline text object you want to edit.
- 2** In the In-Place Text Editor, select the stacked text.
- 3** Right-click in the editor. Click Properties.
- 4** In the Stack Properties dialog box, change settings as needed.
- 5** To set properties for automatic stacking, click Autostack.

6 To save your changes and exit the editor, use one of the following methods:

- Click OK on the toolbar.
- Click in the drawing outside the editor.
- Press CTRL+ENTER.

Draw toolbar



Command line: MTEXT

To unstack text

- 1** Double-click the multiline text object you want to edit.
- 2** In the In-Place Text Editor, select the stacked text.
- 3** Click Stack on the Text Formatting toolbar.
- 4** To save your changes and exit the editor, use one of the following methods:
 - Click OK on the toolbar.
 - Click in the drawing outside the editor.
 - Press CTRL+ENTER.

Draw toolbar



Command line: MTEXT

Create Text with Leaders

You can create text that includes one or more leader lines.

Overview of Text with Leaders

A leader object is a line or a spline with an arrowhead at one end and a multiline text object at the other. In some cases, a short horizontal line, called

a hook line, dogleg, or landing, connects text and feature control frames to the leader line.

The leader line is associated with the multiline text object, so when the text object is relocated, the leader line stretches accordingly.

When associative dimensioning is turned on and object snaps are used to locate the leader arrowhead, the leader is associated with the object to which the arrowhead is attached. If the object is relocated, the arrowhead is relocated, and the leader line stretches accordingly.

You can copy text used elsewhere in a drawing and append a leader to it.

See also:

["Change Text with a Leader" on page 846](#)

Create a Leader Line

A leader object typically consists of an arrowhead, a leader line or curve, and a multiline text object.

You can create a leader line from any point or feature in a drawing and control its appearance as you draw. Leaders can be straight line segments or smooth spline curves. Leader color is controlled by the current dimension line color. Leader scale is controlled by the overall dimension scale set in the current dimension style. The type and size of arrowhead, if one is present, are controlled by the first arrowhead defined in the current dimension style.

LEADER creates complex leader lines that can be made up of more than two line segments. DIMDIAMETER and DIMRADIUS create simple automatic leader lines with two line segments for circles and arcs.



Associate Leaders with Objects

When associative dimensioning is turned on (DIMASSOC), the leader arrowhead can be associated with a location on an object using an object snap. If the object is relocated, the arrowhead remains attached to the object and the leader line stretches, but the multiline text remains in place.

To create a leader with straight lines

- 1 Click Dimension menu ► Leader.
- 2 Press ENTER to display the Leader Settings dialog box and make the following selections:
 - On the Leader Line & Arrow tab, select Straight. Under Number of Points, select No Limit.
 - On the Annotation tab, select MText.
 - Click OK.
- 3 Specify the First leader point and Next points of the leader line.
- 4 Press ENTER to finish selecting leader points.
- 5 Specify the text width.
- 6 Enter the line of text. Press ENTER to enter new lines of text as needed.
- 7 Press ENTER twice to end the command.

After the QLEADER command is completed, the text annotation becomes a multiline text object.

Dimension toolbar



Command line: QLEADER

To create a two-segment spline leader with text

- 1 Click Dimension menu ► Leader.
- 2 Press ENTER to display the Leader Settings dialog box, and make the following selections:
 - On the Annotation tab, select MText.
 - On the Leader Line & Arrow tab, select Spline. Under Number of Points, in the Maximum box, enter **3**. Clear the No Limit option if necessary.
- 3 Click OK.
- 4 Specify the first, second, and third (optional) leader line points.
- 5 Specify the width of the text column.

- 6 Enter the first line of text. To add additional lines, press ENTER once.
- 7 Press ENTER twice to end the command.



Dimension toolbar



Command line: QLEADER

To create multiple leaders from the same annotation

- 1 Select the leader and then the leader arrowhead grip.
- 2 At the Command prompt, enter **c** to select the Copy option.
- 3 Specify the endpoints for the multiple leaders, and then press ENTER.
- 4 To move the end of the new leader to the hook line (dogleg or landing), clear any grips by pressing ESC, select the new leader line, select the leader line grip, and move the grip to the hook line.

Dimension toolbar



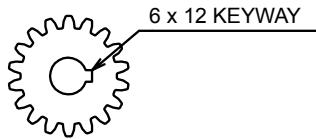
Command line: QLEADER

Add Text to a Leader

There are two ways to create leader text. You can enter text on the command line, or you can create paragraphs of text using the In-Place Text Editor. If you create multiline text, you can apply formatting to individual words or letters. You can also append copies of existing annotation to a leader.

Text is placed automatically at the endpoint of the leader, at the specified offset. You specify this offset in the Modify Dimension Style Manager, Text tab, under Text Placement. Feature control frames also are placed automatically at the endpoint of the leader. Blocks are inserted at the location, scale, and rotation you specify.

Use the Vertical settings in the Modify Dimension Style Manager, Text tab, under Text Placement to align the annotation vertically with the leader. In the following illustration, the vertical justification option used is Above.



To create a box around the text, set the DIMGAP system variable to a negative value. The offset between the box and the text is the absolute value of DIMGAP.

To create a leader using the In-Place Text Editor

- 1 Click Dimension menu ► Leader.
- 2 Specify the First leader point and Next points of the leader.
- 3 Press ENTER to end the leader.
- 4 Specify the text width.
- 5 At the Enter First Line of Annotation prompt, press ENTER.
- 6 In the In-Place Text Editor, enter and format the text.
- 7 To save your changes and exit the editor, use one of the following methods:
 - Click OK on the toolbar.
 - Click in the drawing outside the editor.
 - Press CTRL+ENTER.

The text is appended to the leader endpoint at the offset distance specified in the Dimension Style Manager (DIMSTYLE).

Dimension toolbar



Command line: QLEADER

Import Text from External Files

You can insert TXT or RTF text files created in word processors into your drawing by either importing the text or dragging a file icon from Windows Explorer.

Importing TXT or RTF files from other sources gives you the most flexibility. For example, you can create a text file of standard notes that you include in drawings. The imported text becomes a multiline text object, which you can edit and reformat. Text imported from a TXT file inherits the current text style. Text imported from an RTF file inherits the current text style *name*, but retains its original fonts and format. Imported text files are limited to 32 KB and must have a file extension of .txt or .rtf.

If you drag a text file into a drawing, text width is determined by line breaks and carriage returns in the original document. When you drag an RTF file into a drawing, the text is inserted as an OLE object.

If you use the Clipboard to paste text from another application, the text becomes an OLE object. If you use the Clipboard to paste text from another file, the text is inserted as a block reference, and it retains its original text style.

See also:

["Import OLE Objects"](#)

To import text files

- 1 Click Draw menu ► Text ► Multiline Text.
- 2 Specify opposite corners of a bounding box to define the width of the multiline text object.
- 3 Right-click in the editor. Click Import Text.
The size limit for an imported file is 32 KB.
- 4 In the Select File dialog box, double-click the file you want to import, or select the file. Click Open.
The text is inserted at the cursor location.
- 5 Change the text as needed.
- 6 To save your changes and exit the editor, use one of the following methods:
 - Click OK on the toolbar.
 - Click in the drawing outside the editor.

■ Press CTRL+ENTER.

Draw toolbar



Command line: MTEXT

To insert a text file using the drag-and-drop method

- 1 Open Windows Explorer, but make sure it *does not* fill the screen.
- 2 Display the folder that contains the TXT or RTF file you want.
- 3 Drag the TXT or RTF file icon onto the drawing. TXT files are inserted as multiline text objects using the current text style. RTF files are inserted as OLE objects.

Use Fields in Text

A field is updatable text that is set up to display data that may change during the life cycle of the drawing. When the field is updated, the latest value of the field is displayed.

Insert Fields

A field is text that contains instructions to display data that you expect to change during the life cycle of the drawing.

When a field is updated, the latest data is displayed. For example, the value of the FileName field is the name of the file. If the file name changes, the new file name is displayed when the field is updated.

Fields can be inserted in any kind of text (except tolerances), including text in table cells, attributes, and attribute definitions. When any text command is active, Insert Field is available on the shortcut menu.

Some sheet set fields can be inserted as placeholders. For example, you can insert SheetNumberAndTitle as a placeholder. Later, when the layout is added to a sheet set, the placeholder field displays the correct sheet number and title.

Block placeholder fields can be used in block attribute definitions while you're working in the Block Editor.

A field for which no value is available displays hyphens (----). For example, the Author field, which is set in the Drawing Properties dialog box, may be blank. An invalid field displays pound signs (####). For example, the CurrentSheetName field, which is valid only in paper space, displays pound signs if it is placed in model space.

Change the Appearance of a Field

The field text uses the same text style as the text object in which it is inserted. By default, fields are displayed with a light gray background that is not plotted (FIELDDISPLAY system variable).

Formatting options in the Field dialog box control the appearance of the text that is displayed. The options that are available depend on the type of field. For example, the format for date fields includes options for displaying the day of the week and the time, and the format for named object fields includes capitalization options.

Edit a Field

A field is part of a text object and it can be edited from a text editor. The easiest way to edit a field is to double click the text object that contains the field and then, to display the Field dialog box, double click the field. These operations are available on the shortcut menus as well.

If you no longer want to update a field, you can preserve the value that is currently displayed by converting the field to text.

The field expression, consisting of escape characters and a field code, is shown in the Field dialog box but cannot be edited.

To insert a field in text

- 1 Double-click the text to display the appropriate text editing dialog box.
- 2 Place the cursor where you want the field text to appear and right-click. Click Insert Field.
For keyboard access, press CTRL+F.
- 3 In the Field dialog box, in Field Category, select All or select a category. The fields in the selected category are displayed in the Field Names list.
- 4 In the Field Names list, select a field.
The current value of most fields is displayed in a shaded text box to the right of Field Category. The current value of a date field is displayed in the Examples list.
- 5 Select a format and any other options.

For example, when the NamedObject field is selected, you select a type (for example, layer or textstyle) and a name (for example, 0 for layer or STANDARD for textstyle).

Field Expression displays the expression that underlies the field. The field expression cannot be edited, but you can learn how fields are constructed by viewing this area.

- 6 Click OK to insert the field.

The field displays its current value in the text when the Field dialog box closes.

To insert a field in a table

- 1 Double-click inside a cell in a table to select it for editing.
- 2 Place the cursor where you want the field text to appear and right-click. Click Insert Field.
- 3 In the Field dialog box, select All or select a category.
- 4 In the Field Names list, select a field.
The current value of the field is displayed in a shaded text box to the right of Field Category.
- 5 Select a format and any other option.
- 6 Click OK to insert the field.
The field displays its current value when you move to the next cell.

To use a field to display a property of an object

- 1 Double-click a text object to display the appropriate text editing dialog box.
- 2 Place the cursor where you want the field text to appear and right-click. Click Insert Field.
- 3 In the Field dialog box, in Field Category, select All.
- 4 In the Field Names list, select Object.
- 5 In Object Type, click the Select Object button, and select an object in the drawing.
- 6 In the Field dialog box, in Property, select the property whose value you want to display in the text.
For example, the field could display the radius of a selected circle.
- 7 Select a format for the text.

- 8 Click OK.

The current value for the object's property is displayed in the text.

To insert a sheet set placeholder field

- 1 On the Draw menu, click Block ► Define Attributes.
- 2 In the Attribute Definition dialog box, under Mode, click Preset. Specify any text options.
- 3 Under Attribute, in the Tag box, enter a name for the field.
- 4 To the right of the Value box, click the Insert Field button.
- 5 In the Field dialog box, under Field Category, select SheetSet.
- 6 In the Field Names list, select SheetSetPlaceholder.
- 7 In Placeholder Type, select the type of field that you want.
Temporary Value previews the placeholder field.
- 8 In Format, select a capitalization style for the placeholder field.
When the field is updated, the same style is used for the value of the field.
- 9 (Optional) Click Associate Hyperlink if you want the field to serve as a hyperlink to a location.

NOTE Associate Hyperlink is not available for the ViewportScale field.

- 10 Click OK to exit the Field dialog box.

The placeholder field displays its name when the Field dialog box closes: for example, SheetNumber. Later, when the block is inserted from the Views List tab shortcut menu in the Sheet Set Manager, the field displays a value based on the sheet it is dragged to: for example, its sheet number.

- 11 Click OK in the Attribute Definition dialog box. Specify the location for the field text to appear near the geometry that you have created to use with the Sheet Set Manager as a callout block or a label block.
- 12 Create a block that includes the newly created attribute object along with the geometry for the block.

To format a field value

- 1 Double-click a text object to display the appropriate text editing dialog box.
- 2 Double-click the field you want to format.

The Field dialog box is displayed. If formatting is available for the field, the Field Format button is displayed.

3 Click Field Format.

In the Additional Format dialog box, the current value of the field is displayed. When you select an option, the result is displayed in Preview.

4 Enter a conversion factor to apply to the current value.

For example, to convert inches to millimeters, enter **0.03937**.

5 Enter any text that you want to precede or follow the field value.

For example, enter **mm** for millimeters.

6 Select a decimal separator. Select None or Comma to group thousands.

7 Select an option for suppressing zeros:

■ *Leading*: Suppresses leading zeros in all decimal field values. For example, 0.5000 becomes .5000.

■ *Trailing*: Suppresses trailing zeros in all decimal field values. For example, 12.5000 becomes 12.5, and 30.0000 becomes 30.

■ *0 Feet*: Suppresses the feet portion of a feet-and-inches field value when the distance is less than one foot. For example, 0'-6 1/2" becomes 6 1/2".

■ *0 Inches*: Suppresses the inches portion of a feet-and-inches field value when the distance is an integral number of feet. For example, 1'-0" becomes 1'.

8 Click OK.

In the Field dialog box, the field value is displayed in Preview with the formatting you specified.

9 Click OK.

To edit a field

1 Double-click a text object to display the appropriate text editing dialog box.

2 Double-click the field that you want to edit.

The Field dialog box is displayed.

3 Make any needed changes.

4 Click OK to exit the Field dialog box.

5 Exit the text editor.

Update Fields

You can also set fields to be updated automatically when the drawing is opened, saved, plotted, regenerated, or sent via ETRANSMIT.

Settings on the User Preferences Tab (Options Dialog Box) control whether fields are updated automatically or on demand (FIELDEVAL system variable). The Date field cannot be updated automatically regardless of the setting of FIELDEVAL.

NOTE When the DEMANDLOAD system variable is set to 2, fields cannot be updated until you use FIELD or UPDATEFIELD.

Contextual Fields in Blocks and Xrefs

Some fields are contextual; that is, their value is different depending on which space or layout they reside in. For example, because each layout can have a different page setup attached, the value displayed by the PlotOrientation field can be different in different layouts in the same drawing.

List of contextual fields

CurrentSheetCustom	CurrentSheetTitle
CurrentSheetDescription	DeviceName
CurrentSheetNumber	PaperSize
CurrentSheetNumberAndTitle	PlotDate
CurrentSheetSet	PlotOrientation
CurrentSheetSetCustom	PlotScale
CurrentSheetSetDescription	PageSetupName
CurrentSheetSubSet	PlotStyleTable

For compatibility with previous versions, contextual fields in blocks and xrefs are not updated when you insert them in a drawing; instead, the field displays the last cached value. Therefore, if you want to use a contextual field within a block, for example, a title block, you must insert the field as an attribute. For example, a title block can use the CurrentSheetNumber field as an attribute. When you insert the title block, the field displays the sheet number of the sheet on which the title block is inserted.

Most fields are not contextual and are updated in blocks and xrefs. Fields in xrefs are updated based on the host file, not the source xref. These fields do not have to be placed in attributes. For example, a field that displays the sheet

number of a particular sheet in a sheet set and that updates if that sheet number changes, is a property of the sheet set. When you create the field, you select the SheetSet field name, select the sheet set and the sheet that you want in the tree view, and then select the property SheetNumber for the field value to be displayed. This field displays the sheet number of that sheet, even if you put the field in a block and insert it in another drawing. If the sheet is removed from the sheet set, it no longer has a sheet number, and the field becomes invalid and displays pound signs.

Some sheet set fields can be inserted as placeholders. For example, when you create your own callout blocks and label blocks, you can insert the SheetNumber field as a placeholder. Later, when the block is inserted from the Views List tab shortcut menu in the Sheet Set Manager, the field displays the sheet number of the drawing.

Compatibility with AutoCAD LT and with Previous Releases of AutoCAD

When a drawing with fields is opened in AutoCAD 2004 or earlier, the fields are not updated; they display the value last displayed in the drawing before it was opened. If no changes are made to a field, it is updated normally when it is reopened in a release that supports fields.

Fields are not available in AutoCAD LT. When a drawing with fields is opened in AutoCAD LT, the fields are evaluated based on the setting of the FIELDEVAL system variable in the drawing, but the FIELDEVAL system variable is not accessible.

See also:

["Work with AutoCAD Drawings in AutoCAD LT"](#)

To update a field manually

- 1** Double-click text.
- 2** Select the field to update and right-click. Click Update Field.

To update multiple fields manually

- 1** On the command line, enter **updatefield**.
- 2** At the Select Objects prompt, select the objects that contain the fields you want to update and press ENTER.

All of the fields in the selected objects are updated.

To update fields automatically

- 1** On the command line, enter **fieldeval**.

2 Enter a bitcode that is the sum of any of the following values:

- **0:** Not updated
- **1:** Updated on open
- **2:** Updated on save
- **4:** Updated on plot
- **8:** Updated on use of ETRANSMIT
- **16:** Updated on regeneration

For example, to update fields only when the file is opened, saved, or plotted, enter **7**.

Use Hyperlinks in Fields

The hyperlink works the same way as a hyperlink attached to an object. When the cursor pauses over the text, a hyperlink cursor is displayed, along with a tooltip that describes the hyperlink. Hold down the CTRL key and click to follow the link.

NOTE The Hyperlink field uses an absolute path to a file; the HYPERLINK command can create a hyperlink with a relative path.

The fields that point to sheet and view titles and numbers can be assigned a hyperlink when they are created. When these items are changed or moved in the Sheet Set Manager, the hyperlinks associated with them still jump to the correct location.

Hyperlinks in fields are converted from links to DWG files into links to DWF files when published to a multi-sheet DWF file format.

To add a hyperlink field to text

- 1** Click Draw menu ▶ Text ▶ Multiline Text.
- 2** Place the cursor where you want the hyperlink text to appear.
- 3** Right-click in the editor. Click Insert Field.
- 4** In the Field dialog box, in Field Category, select Linked.
- 5** In Field Names, select Hyperlink, and click Hyperlink.

- 6** In the Insert Hyperlink dialog box, use one of the following methods to specify a location:
 - Under Type the File or Web Page Name, enter the path and name of the file that you want to associate with the hyperlink.
 - Under Browse For, click File, Web Page, or Target. Navigate to the location to which you want to link. Click Open or OK.
- 7** (Optional) In Text to Display, select the default text that is displayed, and enter the link text that you want to appear in the mtext object.
- 8** Click OK to close each dialog box.
- 9** To save your changes and exit the editor, use one of the following methods:
 - Click OK on the toolbar.
 - Click in the drawing outside the editor.
 - Press CTRL+ENTER.

The hyperlink is displayed in the mtext object with the link text that you entered. Use CTRL+click to jump to the hyperlinked location.

Draw toolbar



Command line: MTEXT

Work with Text Styles

When you enter text into your drawing, the current text style determines the text font, size, angle, orientation, and other text characteristics.

Overview of Text Styles

All text in a drawing has a text style associated with it. When you enter text, the program uses the current text style, which sets the font, size, obliquing angle, orientation, and other text characteristics. If you want to create text using a different text style, you can make another text style current. The table shows the settings for the STANDARD text style.

The settings for the current text style are displayed at the prompts on the command line. You can use or modify the current text style or create and load

a new text style. Once you've created a text style, you can modify its characteristics, change its name, or delete it when you no longer need it.

Create and Modify Text Styles

Except for the default STANDARD text style, you must create any text style that you want to use.

Text style names can be up to 255 characters long. They can contain letters, numbers, and the special characters dollar sign (\$), underscore (_), and hyphen (-). If you don't enter a text style name, the text styles are automatically named Style n , where n is a number that starts at 1.

You can modify an existing text style in the Text Style dialog box by changing the settings. You can also update existing text of that text style to reflect the changes.

Certain style settings affect multiline and single-line text objects differently. For example, changing the Upside Down and Backwards options has no effect on multiline text objects. Changing Width Factor and Obliquing has no effect on single-line text.

If you rename an existing text style, any text using the old name assumes the new text style name.

You can remove unreferenced text styles from your drawing with PURGE or by deleting the text styles from the Text Styles dialog box. The STANDARD text style cannot be removed.

Change Text Style

When you change the text style of a multiline text object, the updated settings are applied to the entire object, and some formatting of individual characters might not be retained. The following table describes the effects of text style change on character formatting.

Formatting	Retained?
Bold	No
Color	Yes
Font	No
Height	No
Italic	No
Stacking	Yes
Underlining	Yes

To set the current text style

- On the Styles toolbar, in the Text Style control, click the arrow and select a text style from the list.

Command line: STYLE

Assign Text Fonts

You can assign a text font as part of the text style definition. Several factors depend on the type of text you are working with.

Overview of Assigning Text Fonts

Fonts define the shapes of the text characters that make up each character set. You can use TrueType fonts in addition to compiled SHX fonts.

A single font can be used by more than one text style. If your company has a standard font, you can modify other text style settings to create a set of text styles that use this standard font in different ways. The following illustration shows the same font used by different text styles that use different obliquing settings to define the slant of the text.

You can assign a font to a text style by selecting a font file from the list in the Text Style dialog box.

Use TrueType Fonts

Several factors affect the display of TrueType fonts in a drawing.

TrueType fonts always appear filled in your drawing; however, when you plot, the TEXTFILL system variable controls whether the fonts are filled. By default TEXTFILL is set to 1 to plot the filled-in fonts.

The In-Place Text Editor can display only fonts that are recognized by Microsoft Windows. Because SHX fonts are not recognized by Windows, a TrueType equivalent is supplied in the In-Place Text Editor when you select an SHX or any other non-TrueType font for editing.

See also:

["Set Text Height" on page 838](#)

To assign a TrueType font to a text style

- 1 Click Format menu ▶ Text Style.

- 2 In the Text Style dialog box under Style Name, click New.
- 3 In the New Text Style dialog box, enter a style name for the new text style. Click OK.
- 4 Under Font Name, select a TrueType font from the list. TrueType fonts display a TrueType icon in front of their names.
- 5 To see the effects on different characters, enter a text string in the sample text box that is located to the left of the Preview button. Click Preview.
- 6 To update text of the current style in the drawing, click Apply.
- 7 Click Close.

Styles toolbar



Command line: STYLE

Use Text Fonts for International Work

Several factors affect your choosing, entering, and displaying international text in a drawing.

The program supports the Unicode character-encoding standard. An SHX font encoded using the Unicode standard font can contain many more characters than are defined in your system; therefore, to use a character not directly available from the keyboard, you can enter the escape sequence \U+nnnn, where *nnnn* represents the Unicode hexadecimal value for the character.

Beginning with AutoCAD 2007, all SHX shape fonts are encoded with the Unicode standard with the exception of Big Fonts. When choosing a text font for international work, you can use either a TrueType Font or a Big Font.

Asian Big Font SHX Files

Asian alphabets contain thousands of non-ASCII characters. To support such text, the program provides a special type of shape definition known as a Big Font file. You can set a style to use both regular and Big Font files.

Asian Language Big Fonts Included in the Product

Font File Name	Description
@extfont2.shx	Japanese vertical font (a few characters are rotated to work correctly in vertical text)

Asian Language Big Fonts Included in the Product

Font File Name	Description
bigfont.shx	Japanese font, subset of characters
chineset.shx	Traditional Chinese font
extfont.shx	Japanese extended font, level 1
extfont2.shx	Japanese extended font, level 2
gbcbig.shx	Simplified Chinese font
whgdtxt.shx	Korean font
whgtxt.shx	Korean font
whtgtxt.shx	Korean font
whtmtxt.shx	Korean font

When you specify fonts using **STYLE** Command Line, the assumption is that the first name is the normal font and the second (separated by a comma) is the Big Font. If you enter only one name, it's assumed that it is the normal font and any associated Big Font is removed. By using leading or trailing commas when specifying the font file names, you can change one font without affecting the other, as shown in the following table.

Specifying fonts and Big Fonts on the Command Line

Enter this ...	To specify this ...
[font name],[big font name]	Both normal fonts and Big Fonts
[font name],	Only a normal font (Big Font unchanged)
,[big font name]	Only a Big Font (normal font unchanged)
[font name]	Only a normal font (Big Font, if any, removed)
ENTER (null response)	No change

NOTE Long file names that contain commas as font file names are not accepted. The comma is interpreted as a separator for an SHX font-Big Font pair.

See also:

["Substitute Fonts" on page 836](#)

To assign an Asian-language SHX font to a text style

- 1 Click Format menu ► Text Style.
- 2 In the Text Style dialog box, under Style Name, click New.
- 3 In the New Text Style dialog box, enter a style name for the new text style. Click OK.
- 4 Under Font Name, select the name of an SHX font file, and then select Use Big Font to select an Asian-language big font.
When you select Use Big Font, the Font Style box changes to a Big Font Name box. Only SHX fonts are available for selection, and only Big Font names are shown in the Big Font box.
- 5 To see the effects on different characters, enter a text string in the sample text box that is located to the left of the Preview button. Click Preview.
- 6 To update text of the current style in the drawing, click Apply.
- 7 Click Close.

Styles toolbar



Command line: STYLE

Substitute Fonts

The program accommodates a font that is not currently on your system by substituting another font.

Specify an Alternate Font

If your drawing specifies a font that is not currently on your system, the font designated as your alternate font is automatically substituted. By default, the *simplex.shx* file is used. If you want to specify a different font, enter the alternate font file name by changing the FONTALT system variable. If you use a text style that uses a Big Font, you can map it to another font using the FONTALT system variable. This system variable uses a default font file pair of *txt.shx*, *bigfont.shx*. For more information, see “Use Text Fonts for International Work” on page 834.

In previous releases, you could display PostScript® fonts in the drawing. Because later releases cannot display PostScript fonts, Autodesk has supplied TrueType font equivalents. These PostScript fonts are mapped to the equivalent TrueType fonts in a font mapping file that comes with the program.

Display Proxy Fonts

For third-party or custom SHX fonts that have no TrueType equivalent, one of several different TrueType fonts called proxy fonts is substituted. In the In-Place Text Editor, proxy fonts look different from the fonts they represent to indicate that the proxy fonts are substitutions for the fonts used in the drawing.

Custom SHX fonts do not appear in the Font list on the Text Formatting toolbar. If you want to format characters by assigning one of these fonts, first create a text style that uses the font and then apply that text style to the characters.

Use Font Mapping Files

Sometimes you might want to ensure that your drawing uses only certain fonts, or perhaps you might want to convert the fonts you used to other fonts. You can use any text editor to create font mapping tables for both of these purposes.

You can use these font mapping files to enforce corporate font standards, or to facilitate off-line printing. For example, if you share drawings with consultants, you may want to use a font mapping table to specify what font is substituted when a text object created with another font is encountered. Similarly, you can edit the drawing using quicker-drawing SHX fonts and then switch to more complex fonts for the final plot by setting up a font mapping file that converts each SHX font to an equivalent.

The font mapping file is a plain ASCII text (FMP) file containing one font mapping per line. Each line contains the name of a font file (with no directory name or path) followed by a semicolon (;) and the name of the substitute font file. The substitute file name includes a file extension such as *.ttf*.

For example, you could use the following entry in a font map table to specify that the *times.ttf* TrueType font file be substituted for the *romanc.shx* font file:

romanc.shx;times.ttf

The program comes with a default font mapping file. You can edit this file using any ASCII text editor. You can also specify a different font mapping file in the Options dialog box by using the FONTMAP system variable. The following table shows the font substitution rules used if a font file cannot be located when a drawing is opened.

Font substitution

File extension	First mapping order	Second mapping order	Third mapping order	Fourth mapping order
.ttf	Use font mapping table	Use font defined in text style	Windows substitutes a similar font	
.shx	Use font mapping table	Use font defined in text style	Use FONTALT	Prompt for new font
.pfb	Use font mapping table	Use FONTALT	Prompt for new font	

To specify a font mapping file

- 1 Click Tools menu ► Options.
- 2 On the Files tab, in the list, double-click Text Editor, Dictionary, and Font File Names.
- 3 Double-click Font Mapping File.
The *acad.fmp* file is specified by default.
- 4 To change the font mapping file, double-click the arrow line to open the Select a File dialog box. Select a file and click Open. Click OK.
- 5 At the Command prompt, enter **regen** to convert existing text using the new font mappings.

Command line: OPTIONS

To specify a default alternate font

- 1 At the Command prompt, enter **fontalt**.
- 2 Enter the name of the font file you want to use as the alternative.

Set Text Height

Text height determines the size in drawing units of the letters in the font you are using. Except in TrueType fonts, the value usually represents the size of the uppercase letters.

If you specify a fixed height as part of a text style, the Height prompt is bypassed when you create single-line text. When the height is set to 0 in the text style, you are prompted for the height each time you create single-line text. Set the value to 0 if you want to specify the height as you create text.

TrueType Fonts

For TrueType fonts, the value specified for text height represents the height of a capital letter plus an ascent area reserved for accent marks and other marks used in non-English languages. The relative portion of text height that is assigned to capital letters and ascent characters is determined by the font designer at the time the font is designed; consequently, it varies from font to font.

In addition to the height of a capital letter and the ascent area that make up the text height specified by the user, TrueType fonts have a descent area for portions of characters that extend below the text insertion line, for example, *y, j, p, g, and q*.

When you apply a text height override to all text in the editor, the entire multiline text object is scaled, including its width.

To set text height in a text style

- 1 Click Format menu ► Text Style.
- 2 In the Text Style dialog box, select a style from the Style Name list.
- 3 Under Font, enter the text height (in drawing units) in the Height box.
- 4 To update existing text that uses this text style, click Apply.
- 5 Click Close.

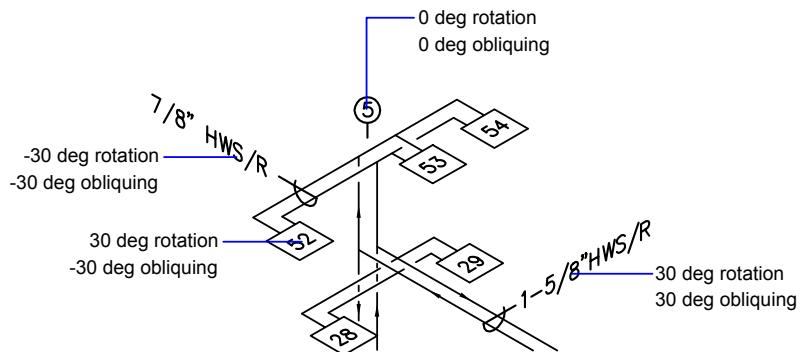
Styles toolbar



Command line: STYLE

Set Text Obliquing Angle

The obliquing angle determines the forward or backward slant of the text. The angle represents the offset from 90 degrees.



Entering a value between -85 and 85 makes the text oblique. A positive obliquing angle slants text to the right. A negative obliquing angle slants text to the left.

To set the obliquing angle in a text style

- 1 Click Format menu ▶ Text Style.
- 2 In the Text Style dialog box, select a text style from the Style Name list.
- 3 Under Effects, enter an angle between -85 and 85 in the Oblique Angle box.
A positive value slants text to the right. A negative value slants text to the left.
- 4 To update existing text that uses this text style, click Apply.
- 5 Click Close.

Styles toolbar

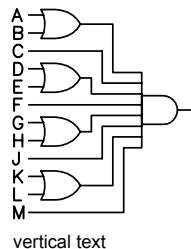


Command line: STYLE

Set Horizontal or Vertical Text Orientation

Lines of text are oriented to be vertical or horizontal. Text can have a vertical orientation only if the associated font supports dual orientation. You can create more than one line of vertical text. Each successive text line is drawn

to the right of the preceding line. The normal rotation angle for vertical text is 270 degrees.



NOTE Vertical orientation is not supported for TrueType fonts and symbols.

To set vertical orientation in a text style

- 1 Click Format menu ► Text Style.
- 2 In the Text Style dialog box, select a text style from the Style Name list.
- 3 Under Effects, select Vertical.
- 4 To update existing text that uses this text style, click Apply.
- 5 Click Close.

Styles toolbar



Command line: STYLE

Change Text

You can change text content, formatting, and properties such as scale and justification.

Overview of Changing Text

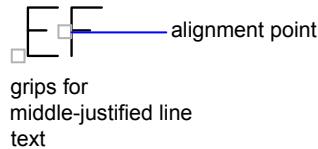
Text, whether created with TEXT, MTEXT, LEADER, or QLEADER, can be modified like any other object. You can move, rotate, erase, and copy it. You can change text properties in the Properties palette.

You can also edit the contents of existing text and create a mirror image of it. The MIRRTEXT system variable controls whether text is also reversed when you mirror objects in your drawing. The procedures for modifying text vary slightly, depending on how the text was created.

Change Single-Line Text

You can change single-line text with DDEDIT and PROPERTIES. Use DDEDIT when you need to change only the content of the text, not the formatting or properties of the text object. Use PROPERTIES when you want to change content, text style, location, orientation, size, justification, and other properties.

Text objects also have grips for moving, scaling, and rotating. A text object has grips at the lower-left corner of the baseline and at the alignment point.



The effect of a command depends on which grip you choose.

To edit single-line text

- 1 Click Modify menu > Object > Text > Edit.
- 2 Select a single-line text object.
- 3 In the in-place editor, enter the new text.
- 4 Press ENTER.
- 5 Select another text object to edit, or press ENTER to end the command.

Command line: DDEDIT

To modify properties of single-line text objects

- 1 Select a single-line text object.
- 2 Right-click the selected object. Click Properties.
- 3 In the Properties palette, enter any new text, and then change formatting and other properties as needed.

Standard toolbar



Command line: PROPERTIES

Change Multiline Text

You can change the location and content of multiline text objects with the Properties palette, the In-Place Text Editor, and grips. You can easily find and replace text with the FIND command.

After you create multiline text, you can use the Properties palette to change

- Text style assignment
- Justification
- Width
- Rotation
- Line spacing

In addition, you can use the In-Place Text Editor to modify individual formatting, such as boldface and underlining, and to change the width of the multiline text object.

Change Text Location

You can use grips to move multiline text or to resize the line width. A multiline text object has grips at the four corners of the text boundary and, in some cases, at the justification point.

If you use the Properties palette to move multiline text, you can edit content and change properties at the same time.

Commands such as 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.

Find and Replace Text

To search for and replace text, use FIND. Replacement is based on text content only; character formatting and text properties are not changed.

With FIND, you can use wild-card characters in your search.

Character	Definition
# (Pound)	Matches any numeric digit
@ (At)	Matches any alphabetic character
. (Period)	Matches any nonalphanumeric character
* (Asterisk)	Matches any string and can be used anywhere in the search string
? (Question mark)	Matches any single character; for example, ?BC matches ABC, 3BC, and so on
~ (Tilde)	Matches anything but the pattern; for example, ~*AB* matches all strings that don't contain AB
[]	Matches any one of the characters enclosed; for example, [AB]C matches AC and BC
[~]	Matches any character not enclosed; for example, [~AB]C matches XC but not AC
[-]	Specifies a range for a single character; for example, [A-G]C matches AC, BC, and so on up to GC, but not HC
` (Reverse quote)	Reads the next character literally; for example, `~AB matches ~AB

To search for drawing files that contain a specific word or phrase, use the Search tool in Microsoft® Windows® . You can use the Search tool to find all textual data except text in tables and fields, and xrefs within drawing files.

See also:

[“Work with Text Styles” on page 831](#)

[“Control the Display of Polylines, Hatches, Gradient Fills, Lineweights, and Text” on page 323](#)

To change multiline text

- 1 Select a multiline text object.
- 2 Right-click the selected object. Click Properties.

- 3** In the Properties palette, enter any new text and change formatting and other settings as needed.

Standard toolbar



Command line: PROPERTIES

To find specified text in a drawing

- 1** Click Edit menu ► Find.
- 2** In Find Text String, enter the text you want to find.
- 3** In Search In, select Entire Drawing or click the Select Objects button to select one or more text objects.
- 4** Click Options to specify the types of text to include in the search, whether to find whole words only, and whether to match the case of the specified text.
- 5** Click Find.
The Context area displays the found text with its surrounding context.
- 6** To zoom to the found text in the drawing, click Zoom To.
- 7** Click Close.

Command line: FIND

To replace text

- 1** Click Edit menu ► Find.
- 2** In Find Text String, enter the text you want to find.
- 3** In Search In, select Entire Drawing or click the Select Objects button to select one or more text objects.
- 4** Click Options to specify the types of text to include in the search, whether to find whole words only, and whether to match the case of the specified text.
- 5** In Replace With, enter the text with which you want to replace the found text.
- 6** Click Find.
The Context area displays the found text with its surrounding context.

7 Use *one* of the following methods:

- To replace only the found instance of the text string, click Replace.
- To replace all instances of the text in Find Text String, click Replace All.

The replacement is confirmed in the status area at the bottom of the dialog box.

8 Click Close.

Command line: FIND

To change the width of a multiline text object

1 Double-click the multiline text object.

2 In the In-Place Text Editor, use one of the following methods:

- Move the cursor over the right end of the ruler until the cursor changes to a double arrow. As you drag to the right to stretch the ruler, a tooltip displays the width. Release to set a new width.
 - Right-click the bottom of the ruler. Click Set Mtext Width. In the dialog box, enter the width in drawing units.
- 3** To save your changes and exit the editor, use one of the following methods:
- Click OK on the toolbar.
 - Click in the drawing outside the editor.
 - Press CTRL+ENTER.

Change Text with a Leader

You can change both the text component and the leader geometry component of leader objects.

To resize a leader, stretch or scale it. Stretching modifies the offset between the leader endpoint and its annotation. Scaling updates the scale of the selected object only. For example, if you scale the leader, the annotation stays in the same position relative to the leader endpoint but is not scaled itself. Leaders can function as edges for trimming and extending, but they cannot be trimmed or extended.

To edit leader text

- 1 Double-click the text you want to edit.

The In-Place Text Editor is displayed for both single-line text and multiline text. The Text Formatting toolbar is not available for single-line text.

- 2 Edit the text.

You can also select the text and right-click. Click Properties. Edit the text in the Properties palette.

Any modification to the annotation that changes the annotation position or its attachment point affects the leader's endpoint position. Rotating the annotation causes the leader hook line (if any) to rotate.

Command line: DDEDIT

Change Text Scale and Justification

You can change the scale of one or more text objects, attributes, and attribute definitions, or their insertion points, simultaneously without changing the location of the objects.

Change the Scale of Multiple Text Objects

A drawing may contain hundreds of text objects that need to be scaled, and it would be tedious to scale them individually. Use SCALETEXT to change the scale of one or more text objects such as text, multiline text, and attributes. You can specify a relative scale factor or an absolute text height, or you can scale selected text to match the height of existing text. Each text object is scaled using the same scale factor, and it maintains its current location.

Convert Text Height Between Model Space and Paper Space

The SPACETRANS command converts distances between model space units and paper space units. By using SPACETRANS transparently, you can provide commands with distance entries relative to another space. For example, you may want to create a text object in model space that matches the height of other text in a layout. From model space, you could enter

Command: **text**

Specify start point of text or [Justify/Style]: **1,3**

Specify height <0.375>: '**spacetrans**

>>Specify paper space distance <1.000>: **3/8**

Resuming TEXT command

Specify height <0.375>: **1.173**

When the command is complete, a text object is created in model space with a height of 1.173, which appears as 3/8 when viewed from a layout.

For more information about entering commands transparently, see “Enter Commands on the Command Line” on page 34.

Change the Justification of Text Objects Without Changing Their Location

Use JUSTIFYTEXT to redefine the insertion point of text without moving the text. For example, a table or schedule may contain text that is located correctly but each text object in the table should be right-justified instead of left-justified for future entries or modifications.

Check Spelling

You can check the spelling of all text in your drawing, including

- Single-line text
- Multiline text
- Text within attribute values
- Text within block references and their associated block definitions
- Text within nested blocks

Spelling is checked only in objects in the current selection set. If you enter the All option when selecting objects, spelling is checked in all objects in model space and in all layouts. Spelling is not checked in block definitions of unselected block references or in text within dimensions.

Switch Dictionaries

The program contains several main dictionaries, which are available in different languages. The main dictionaries use a standard word list that you can customize.

During a spelling check, the words in the drawing are matched to the words in the current main dictionary. Any words you add are stored in the custom dictionary that is current at the time of the spelling check. For example, you can add proper names so they are no longer identified as misspelled words.

To check spelling in another language, you can change to a different main dictionary. You can also create any number of custom dictionaries and switch to them as needed.

You can change dictionaries in a dialog box or by specifying the dictionary name in the DCTMAIN or DCTCUST system variable. For a list of the dictionary file names, see DCTMAIN.

Create and Edit Custom Dictionaries

A custom dictionary is a list of spelling exceptions that you have identified. The files that contain them can be identified by the .cus file extension. You can use any ASCII text editor to add and delete words or to combine dictionaries. If a word is preceded by a tilde (~), the word is always flagged as incorrect.

You can also create and edit dictionaries from inside the program.

To check spelling

- 1 Click Tools menu ► Spelling.
- 2 Select the text objects you want to check, or enter **all** to select all text objects.
- 3 If no misspelled words are found , a message is displayed. If a misspelling is found , the Check Spelling dialog box identifies the misspelled word.
- 4 Do *one* of the following:
 - To correct a word, select an alternate word from the Suggestions list or type a word in the Suggestions box. Click Change or Change All.
 - To leave a word unchanged, click Ignore or Ignore All.
 - To leave a word unchanged and add it to the custom dictionary, click Add. (This option is not available unless a custom dictionary is specified.)
- 5 Repeat step 4 for each misspelled word. Click OK or Cancel to exit.

Command line: SPELL

To switch dictionaries while checking spelling

- 1 Click Tools menu ► Spelling.
- 2 Select the text that you want to check and press ENTER.
- 3 In the Check Spelling dialog box, click Change Dictionaries.
- 4 Do *one* of the following:
 - To change the main dictionary, select a dictionary from the Main Dictionary list.

- To change the custom dictionary, select or enter a name under Custom Dictionary.
 - To select from a list of dictionaries, click Browse.
- 5 Click Apply and Close.

Command line: SPELL

To create or edit custom dictionaries during a spelling check

- 1 Click Tools menu > Spelling.
- 2 Select the text that you want to check and press ENTER.
- 3 In the Check Spelling dialog box, click Change Dictionaries.
- 4 In the Change Dictionaries dialog box, under Custom Dictionary, specify the dictionary you want to edit using one of these methods:
 - To specify a dictionary, enter its name, using the .cus file extension.
 - To select from a list of folders, click Browse.
 - To create a new custom dictionary, enter a new name. Use the .cus file extension.
- 5 To add a word, enter it under Custom Dictionary Words. Click Add.
- 6 To remove a word, select it from the list. Click Delete.
- 7 Click OK.

Command line: SPELL

Use an Alternate Text Editor

The default text editor is the In-Place Text Editor, but you can elect to use any alternate editor that saves files in ASCII format.

Overview of Using an Alternate Text Editor

The default text editor is the In-Place Text Editor, but you can elect to use an alternate editor by specifying the editor with the MTEXTED system variable. You can use any text editor, such as Microsoft Notepad, that saves files in ASCII format.

If you use an alternate text editor for multiline text, you specify the properties of the multiline text object on the command line first. Then the text editor opens for entering text. When you close the text editor, the text is inserted within the width limit you specified.

If you use an alternate editor, you must enter special codes to apply formatting.

To edit text using an alternate text editor, use the same format codes. To avoid losing format information when you make changes to the text, use the same text editor you used to create the text.

Format Multiline Text in an Alternate Text Editor

If you use an alternate text editor, you apply formatting by entering format codes. You can underline text, add a line over text, and create stacked text. You can also change color, font, and text height. You can change the spaces between text characters or increase the width of the characters themselves. To apply formatting, use the format codes shown in the following table.

Format codes for paragraphs

Format code	Purpose	Enter this ...	To produce this ...
\0... \o	Turns overline on and off	Autodesk \OAutoCAD\o	Autodesk <u>AutoCAD</u>
\L... \l	Turns underline on and off	Autodesk \LAutoCAD\l	Autodesk <u>AutoCAD</u>
\~	Inserts a nonbreaking space	Autodesk AutoCAD\~LT	Autodesk AutoCAD LT
\\\	Inserts a backslash	Autodesk \\AutoCAD	Autodesk \AutoCAD
\{...\}	Inserts an opening and closing brace	Autodesk \{AutoCAD\}	Autodesk {AutoCAD}
\Cvalue;	Changes to the specified color	Autodesk \C2;AutoCAD	Autodesk AutoCAD

Format codes for paragraphs

Format code	Purpose	Enter this ...	To produce this ...
\File name;	Changes to the specified font file	Autodesk \Ftimes; AutoCAD	Autodesk AutoCAD
\Hvalue;	Changes to the text height specified in drawing units	Autodesk \H2;AutoCAD	Autodesk AutoCAD
\Hvaluex;	Changes the text height to a multiple of the current text height	Autodesk \H3x;AutoCAD	Autodesk AutoCAD
\S...^...;	Stacks the subsequent text at the /, #, or ^ symbol	1.000\S+0.010^-0.000; at the /, #, or ^ symbol	+0.010 1.000 -0.000
\Tvalue;	Adjusts the space between characters, from .75 to 4 times	\T2;Autodesk	A u t o d e s k
\Qangle;	Changes obliquing angle	\Q20;Autodesk	Autodesk
\Wvalue;	Changes width factor to produce wide text	\W2;Autodesk	Autodesk
\A	Sets the alignment value; valid values: 0, 1, 2 (bottom, center, top)	\A1;1\S1/2	1½
\P	Ends paragraph	Autodesk\PAutoCAD	Autodesk AutoCAD

Braces can be nested up to eight levels deep.

You can also use control codes to add special characters, such as tolerance or dimensioning symbols. See MTEXT.

Example: Formatting Text in an Alternate Text Editor

This example describes how the text in the following illustration was created.

Big text

over text/ under text

Baseline: $1\frac{1}{2}$

Center: $1\frac{1}{2}$

Topline: $1\frac{1}{2}$

Tolerances: $1.000^{+0.010}_{-0.000}$

Architectural: $9-11/16"$

Each line below was entered in an alternate text editor:

```
{\H1.5x; Big text} \A2; over text\A1;/\A0; under text}\P  
\A0;Baseline: 1 \S1/2;}\P  
\A1;Center: 1 \S1/2;}\P  
\A2;Topline: 1 \S1/2;}\P  
\Tolerances: \A1;1.000\H.75x;\S+0.010^-0.000;}\P  
\Architectural: 9-{\H.666x;\A2;11\A1;/\A0;16}\A2;"}\P
```

To specify an alternate text editor

- 1** At the Command prompt, enter **mtexted**.
- 2** At the prompt do *one* of the following:
 - Enter the path and name of the executable file for the ASCII text editor that you want to use to create or edit multiline text.
 - Enter **internal** to restore the text editor.

To create multiline text in an alternate text editor

- 1 To specify a text editor, at the Command prompt, enter **mtexted**. Then enter the path of the editor you want to use.
- 2 Click Draw menu > Text > Multiline Text.
- 3 Specify the first corner of the multiline text boundary rectangle.
- 4 Specify the opposite corner of the multiline text boundary rectangle.
- 5 In the text editor, enter the text. Enter \P to end a paragraph and start a new paragraph on the next line. (Be sure to capitalize the P.)
- 6 When your text entry is complete, save the changes and exit the text editor.

Draw toolbar



Command line: MTEXT

Tables

27

A table is a rectangular array of cells that contain annotation, primarily text but also blocks. Tables appear in many different forms on many of the sheets that make up drawing sets. In the AEC industry, tables are often referred to as “schedules” and contain information about the materials needed for the construction of the building being designed. In the manufacturing industry, they are often referred to as “BOM” (bills of materials).

The table object creates a table of any size that can be used for any purpose, including as a list or index to a set of drawing sheets to be published.

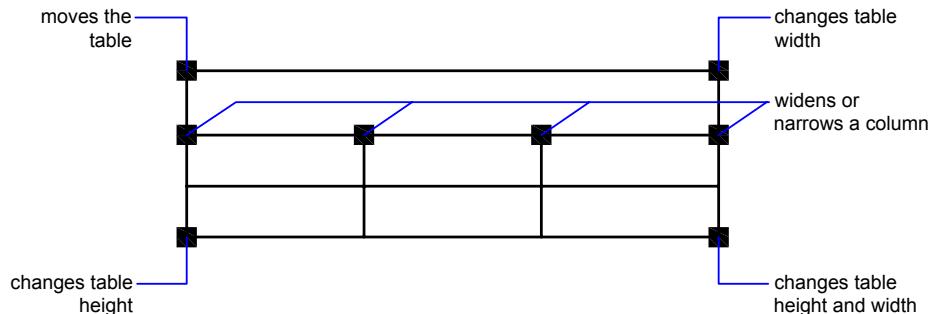
In this chapter

- Create and Modify Tables
- Work with Table Styles
- Add Text and Blocks to Tables
- Use Formulas in Table Cells

Create and Modify Tables

A table is an object that contains data in rows and columns. When you create a table object, you first create an empty table, and then you add content to the cells in the table.

After the table has been created, you can click any gridline on the table to select it and then modify it by using the Properties palette or grips.



When you change the height or width of the table, the rows or columns change proportionally. When you change the width of a column, the table widens or narrows to accommodate the change. To maintain the table width, press CTRL while you use a column grip.

Modify a Table Cell

You can click inside a cell to select it. Grips are displayed in the middle of the cell borders. Click inside another cell to move selection to that cell. Drag the grips on a cell to make the cell and its column or row larger or smaller.

To select more than one cell, click and drag over several cells. You can also hold down SHIFT and click inside another cell to select those two cells and all the cells between them.

With a cell selected, right-click and use the options on the shortcut menu to insert or delete columns and rows, combine adjacent cells, or make other changes. When cells are selected, you can use CTRL+Y to repeat the last action, including changes made in the Property palette.

Add a Table to a Tool Palette

When you add a table to a tool palette, the table properties (for example, table style and number of rows and columns) and the cell property overrides (for

example, alignment and border linewidth) are stored in the tool definition. The text or block content and character formatting is ignored.

See also:

["Add Text and Blocks to Tables" on page 864](#)

To create a table

- 1 Click Draw menu ▶ Table.
- 2 In the Insert Table dialog box, select a table style from the list, or click the [...] button to create a new table style.
- 3 Select an insertion method:
 - Specify an insertion point for the table.
 - Specify a window for the table to fit.
- 4 Set the number of columns and the column width.
If you used the window insertion method, you can select the number of columns or the column width, but not both.
- 5 Set the number of rows and the row height.
If you used the window insertion method, the number of rows is determined by the size of the window you specified and the row height.
- 6 Click OK.

To use grips to modify a table

- 1 Click a gridline to select the table.
- 2 Use one of the following grips:
 - **Upper-left grip.** Moves the table.
 - **Upper-right grip.** Changes the width of the table and all columns proportionally.
 - **Lower-left grip.** Changes the height of the table and all rows proportionally.
 - **Lower-right grip.** Changes both the height and the width of the table and changes the columns and rows proportionally.
 - **Column grip** (at the top of the column head row). Changes the width of the column to the left of the grip and widens or narrows the table to accommodate the change.

- **CTRL+ a column grip.** Widens or narrows adjacent columns without changing the width of the table.

The minimum column width is the width of a single character. The minimum row height for an empty table is the height of a line of text plus the cell margins.

- 3 Press ESC to remove selection.

To use grips to modify cells in a table

- 1 Select one or more cells in a table that you want to change using one of the following methods:
 - Click inside a cell.
 - Hold down SHIFT and click inside another cell to select those two cells and all the cells between them.
 - Click inside the selected cell, drag to the cells you want to select, and release.
- 2 To change the row height of the selected cell, drag the top or bottom grip.
If more than one cell is selected, the row height changes equally for each row.
- 3 To change the column width of the selected cell, drag the left or right grip.
If more than one cell is selected, the column width changes equally for each column.
- 4 To merge selected cells, right-click. Click Merge Cells.
If cells in more than one row or column are selected, you can merge by row or by column.
- 5 Press ESC to remove selection.

To use the Properties palette to modify a table

- 1 Click a gridline to select the table.
- 2 Click Tools menu ➤ Properties.
- 3 In the Properties palette, click the value you want to change and enter or select a new value.
The property is changed in the selected table.

- 4 Move the cursor outside the Properties palette, and press ESC to remove selection.

To change column width or row height in a table

- 1 Click inside a table cell in the column or row you want to change.
Hold down SHIFT and click inside another cell to select those two cells and all the cells between them.
- 2 Click Tools menu ▶ Properties.
- 3 In the Properties palette, under Cell, click the cell width or the cell height value and enter a new value.
- 4 Press ESC to remove selection.

To add columns or rows to a table

- 1 Click inside a table cell where you want to add a column or a row.
You can select more than one cell to add more than one column or row.
- 2 Right-click and use one of the following options:
 - **Insert Column ▶ Right.** Inserts a column to the right of the selected cell.
 - **Insert Column ▶ Left.** Inserts a column to the left of the selected cell.
 - **Insert Row ▶ Above.** Inserts a row above the selected cell.
 - **Insert Row ▶ Below.** Inserts a row below the selected cell.
- 3 Press ESC to remove selection.

To merge cells in a table

- 1 Select the cells in a table that you want to merge using one of the following methods:
 - Select a cell, and hold down SHIFT and click inside another cell to select those two cells and all the cells between them.
 - Click inside a selected cell, drag to the cells you want to select, and release.

The resulting merged cell must be rectangular.

- 2 Right-click. Click Merge Cells. If you want to create more than one merged cell, use one of the following options:
 - **By Row:** Merges the cells horizontally by removing the vertical gridlines and leaving the horizontal gridlines intact.
 - **By Column:** Merges the cells vertically by removing the horizontal gridlines and leaving the vertical gridlines intact.
- 3 Start typing to enter text in the new merged cell, or press ESC to remove selection.

To delete columns or rows in a table

- 1 Click inside a table cell in the column or row that you want to delete. Hold down SHIFT and click inside another cell to select those two cells and all the cells between them.
- 2 Right-click and use one of the following options:
 - **Delete Columns.** Deletes the columns you specified.
 - **Delete Rows.** Deletes the rows you specified.
- 3 Press ESC to remove selection.

To export a table

- 1 At the Command prompt, enter **tableexport**.
- 2 Select the table that you want to export.
A standard file selection dialog box is displayed.
- 3 Enter a file name and select a location for the file.
Table data is exported in the comma-separated (CSV) file format. All table and text formatting is lost.

Command line: With a table selected, right-click. Click Export.

Work with Table Styles

The appearance of the table is controlled by its table style. You can use the default table style, STANDARD, or create your own table styles.

The table style can specify formatting for title, column head, and data rows. In the STANDARD table style, for example, the first row is a title row that

consists of a row of merged cells with text that is centered. The second row is for column heads, and all the other rows are data rows.

The table style can specify different justification and appearance for the text and gridlines in each type of row. For example, a table style can specify larger text for the title row or center alignment for the column heads row and left alignment for the data rows.

The table can read from top to bottom or from the bottom up. The number of columns and rows is almost unlimited.

The border properties in a table style control the display of the gridlines that divide the table into cells. The borders of the title row, the column heads row, and the data rows can have different linewidth and color and can be displayed or not displayed. The preview image in the Table Style dialog box updates as you select border options.

The appearance of text in the cells of the table is controlled by the text style that is specified in the current table style. You can use any text style in the drawing or create a new one. You can also use DesignCenter to copy table styles from other drawings.

You can define the data and formatting for title, column head, or data rows. You can also overwrite the data and formatting for specific cells. For example, you could set the formatting for all column heading rows to display text in uppercase, and then select a single table cell to display text in lower case. 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 831](#)

To define or modify a table style

- 1 Click Format menu ► Table Style.
- 2 In the Table Style dialog box, click New.
- 3 In the Create New Table Style dialog box, enter a name for the new table style, select a table style in Start With to provide default settings for the new table style, and click Continue.
- 4 In the New Table Style dialog box, set the following options for the whole table:
 - **General.** Set the table direction by selecting Down or Up. Up creates a table that reads from the bottom up; the title row and the column heads are at the bottom of the table.

■ **Cell Margins.** Enter a value for the vertical and horizontal spacing between the border of the cell and the cell content. The default setting is one-third of the text height in the data rows, and the maximum is the text height in the data rows.

- 5 To set the appearance of the data cells, the column head cells, or the title cell, click the appropriate tab.
- 6 If you want to include a title row or a header row, on the Title tab or the Column Heads tab (as appropriate), select or clear the following options:

■ **Include Title Row.** When this option is selected, the first row in the table is all one cell that has the appearance that you set on the Title tab.

■ **Include Header Row.** When this option is selected, the first row in each column is a column heads row that has the appearance that you set on the Column Heads tab.

When both options are selected, the first row of the table is a title row, and the second row is a column heads row.

- 7 On any of the tabs, use the following cell properties options to control the appearance of the cell contents:

■ **Text Style.** Select a text style, or click the [...] button to open the Text Style dialog box and create a new text style.

■ **Text Height.** Enter a height for the text. This option is available only when the selected text style has a text height of 0. (The default text style, STANDARD, has a 0 text height.) If the selected text style specifies a fixed text height, this option is unavailable.

■ **Text Color.** Select a color, or click Select Color to display the “Select Color Dialog Box”.

■ **Fill Color.** Select None or a background color, or click Select Color to display the “Select Color Dialog Box”.

■ **Alignment.** Specify an alignment for cell contents. Center refers to horizontal alignment; Middle refers to vertical alignment.

■ **Format.** Set the data type and formatting for the Data, Column Heading, or Title rows in a table. Click the [...] button to display the Table Cell Format Dialog Box, where you can further define formatting options.

- 8** Use the following border properties options to control the appearance of the gridlines:
- **Border display buttons.** Click a button to apply linewidth and color properties to all borders of the cell, the outside border, the inside borders (not available on the Title tab), no borders, or the bottom border. The preview in the dialog box updates to show the effect.
 - **Grid Lineweight.** Enter the linewidth to be used for borders that are displayed. If you use a heavy linewidth, you may have to change the cell margins so that the text can be seen.
 - **Grid Color.** Select a color to be used for borders that are displayed, or click Select Color to display the “Select Color Dialog Box”.
- 9** Click another tab to continue defining the table style, or click OK to exit the dialog box.

To apply a new table style to a table

- 1 Click a gridline to select the table.
- 2 Click Tools menu ▶ Properties.
- 3 In the Properties palette, under Table, click the Table Style value and select a table style.

The new table style is applied to the table.

NOTE If the previous table style had a title row and the new one does not, the title text is placed in the first cell of the table, and the other cells in the first row are left blank.

- 4 Press ESC twice to remove selection.

To change the table style that is applied to new tables

- 1 Click Format menu ▶ Table Style.
- 2 In the Table Style dialog box, select a table style. Click Set Current.
- 3 Click Close.

The current table style is applied to new tables that you create.

To define the data type and formatting of a table row

- 1 Click Format menu ▶ Table Style.

- 2** In the Table Style dialog box, do one of the following:
 - Click New. In the Create New Table Style dialog box, enter a name for the new table style. In the Start With box, select a table style to provide default settings for the new table style. Click Continue.
 - Click Modify.
- 3** In the New Table Style dialog box or Modify Table Style dialog box, click the Data, Column Head, or Title tab.
- 4** In the Cell Properties section, click the ellipses button [...] to the right of Format.
- 5** In the Table Cell Format dialog box, set the data type, format, and other options for the selected table row.
- 6** Click OK.

To define the data type and formatting of selected table cells

- 1** In a table, click the table cells where you want to redefine data and formatting.
- 2** Right-click, and on the shortcut menu, click Format.
- 3** In the Table Cell Format dialog box, set the data type, format, and other options for the selected table cells.
- 4** Enter data in the selected table cells. The data type and format you chose will determine how the data is displayed.
- 5** Click OK.

Add Text and Blocks to Tables

The data in a table cell can be text or a block.

When the table is created, the first cell is highlighted, the Text Formatting toolbar is displayed, and you can begin entering text. The row height of the cell increases to accommodate the number of lines of text. To move to the next cell, press TAB, or use the arrow keys to move left, right, up, and down.

When you insert a block into a table cell, either the block can be automatically fit to the size of the cell, or the cell can adjust to accommodate the size of the block.

Inside the cell, the arrow keys move the cursor. Use the toolbar and the shortcut menu to format text, import text, or make other changes to the text in the cell.

See also:

- “Use Fields in Text” on page 823
- “Create Multiline Text” on page 798

To enter text in a table

- 1 Click inside a table cell, and start to enter text.
The Text Formatting toolbar is displayed.
- 2 Use the arrow keys to move the cursor in text inside a cell.
- 3 To create a line break within a cell, press ALT+ENTER .
- 4 To override the text style specified in the table style, click the arrow next to the Text Style control on the toolbar and select a new text style.
The text style you select is applied to the text in the cell and any new text that you enter in the cell.
- 5 To override the formatting in the current text style, first select text as follows:
 - To select one or more characters, click and drag the pointing device over the characters.
 - To select a word, double-click the word.
 - To select all the text in the cell, triple-click in the cell. (You can also right-click. Click Select All.)
- 6 On the toolbar, make format changes as follows:
 - To change the font of the selected text, select a font from the list.
 - To change the height of the selected text, enter a new value in the Height box.
 - To format text in a TrueType font with boldface or italics, or to create underlined text for any font, click the corresponding button on the toolbar. SHX fonts do *not* support boldface or italics.
 - To apply color to selected text, choose a color from the Color list. Click Other to display the “Select Color Dialog Box”.
- 7 Use the keyboard to move from cell to cell:
 - Press TAB to move to the next cell. In the last cell of the table, press TAB to add a new row.
 - Press SHIFT+TAB to move to the previous cell.

- When the cursor is at the beginning or end of text in a cell, use the arrow keys to move the cursor to adjacent cells. You can also use CTRL+ an arrow key.
 - When text is highlighted in a cell, press an arrow key to remove selection and move the cursor to the beginning or end of the text in the cell.
 - Press ENTER to move down one cell.
- 8** To save your changes and exit, click OK on the toolbar or press CTRL+ENTER .

To change the properties of cells in a table

- 1 Click inside the table cell you want to change.
Hold down SHIFT and click inside another cell to select those two cells and all the cells between them.
- 2 Use one of the following methods:
 - To change one or more properties, in the Properties palette, click the value you want to change and enter or select a new value.
 - To restore the default properties, right-click. Click Remove Property Overrides.

To copy the properties of a cell to other cells

- 1 Click inside the table cell whose properties you want to copy.
- 2 (Optional) To view the current properties of the selected table cell, press CTRL+1 to open the Properties palette.
All the properties of the cell are copied except the cell type: text or block.
- 3 Right-click. Click Match Cell.
The cursor changes to a paintbrush.
- 4 To copy the properties to another table cell in the drawing, click inside the cell.
- 5 Right-click or press ESC to stop copying properties.

To change the linewidth or color of the borders of table cells

- 1 Click inside the table cell you want to change.

Hold down SHIFT and click inside another cell to select those two cells and all the cells between them.

- 2 Right-click. Click Cell Borders.
- 3 In the Cell Border Properties dialog box, select a linewidth and a color. Use BYBLOCK to set the border properties to match the settings in the table style that has been applied to the table.
- 4 Click one of the border type buttons to specify which borders of the cell to modify, or select a border in the preview image.
- 5 Click OK.
- 6 Move the cursor outside the Properties palette, and press ESC to remove selection, or select another cell.

To edit text in a table cell

- 1 Double-click inside the cell whose text you want to edit, or select the cell and click Edit Cell on the shortcut menu.
- 2 Use the Text Formatting toolbar or the shortcut menu to make changes.
- 3 To save the changes and exit, click OK on the toolbar, press CTRL+ENTER, or click outside the cell.
- 4 To remove selection from the table, press ESC.

To insert a block in a table cell

- 1 Click inside the table cell to select it and right-click. Click Insert Block.
 - 2 In the Insert dialog box, select a block from the list of blocks in the drawing, or click Browse to find a block in another drawing.
 - 3 Specify the following properties for the block:
 - **Cell Alignment.** Specifies alignment for the block in the table cell. The block is middle-, top-, or bottom-aligned with respect to the top and bottom borders of the cell. The block is center-, left-, or right-aligned with respect to the left and right borders of the cell.
 - **Scale.** Specifies the scale for the block reference. Enter a value or select AutoFit to scale the block to fit in the selected cell.
 - **Rotation Angle.** Specifies a rotation angle for the block.
 - 4 Click OK.
- If the block has attributes attached, the Edit Attributes dialog box is displayed.

To insert a field in a table cell

- 1 Double-click inside the table cell.
- 2 Right-click. Click Insert Field, or press CTRL+F.
- 3 In the Field dialog box, select a category in the Field Category list to display the field names in that category.
- 4 Select a field.
- 5 Select the format or other options available for that field.
- 6 Click OK.

Use Formulas in Table Cells

Table cells can contain formulas that do calculations using the values in other table cells. With a table cell selected, you can insert formulas from the shortcut menu. You can also open the In-Place Text Editor and enter a formula in a table cell manually.

Insert a Formula

In formulas, cells are referred to by their column letter and row number. For example, the cell at top left in the table is A1. Merged cells use the number of what would be the top-left cell. A range of cells is defined by the first and last cells, with a colon between them. For example, the range A5:C10 includes cells in rows 5 through 10 in columns A, B, and C.

A formula must start with an equal sign (=). The formulas for sum, average, and count ignore empty cells and cells that do not resolve to a numeric value. Other formulas display an error (#) if any cell in the arithmetic expression is empty or contains nonnumeric data.

Use the Cell option on the shortcut menu to select a cell in another table in the same drawing. When you have selected the cell, the In-Place Text Editor opens so you can enter the rest of the formula.

Copy a Formula

When you copy a formula to another cell in the table, the range changes to reflect the new location. For example, if the formula in A10 sums A1 through A9, when you copy it to B10, the range of cells changes so that it sums B1 through B9.

If you don't want a cell address to change when you copy and paste the formula, add a dollar sign (\$) to the column or row part of the address. For

example, if you enter \$A10, the column stays the same and the row changes. If you enter \$A\$10, both column and row stay the same.

Customize Display of Column Letters and Row Numbers

By default, the In-Place Text Editor displays column letters and row numbers when a table cell is selected for editing. Use the TABLEINDICATOR system variable to turn this display on and off. To set a new background color, select a table, right-click, and click Table Indicator Color on the shortcut menu. The text color, size, and style and the line color are controlled by the settings for column heads in the current table style.

See also:

["Use Fields in Text" on page 823](#)

To sum the values in a range of table cells

1 Select the table cell where you want to place the formula by clicking inside it.

2 Right-click. Click Insert Formula ➤ Sum.

The following prompt is displayed:

Select first corner of table range:

3 Click inside the first cell in the range.

The following prompt is displayed:

Select second corner of table range:

4 Click inside the last cell in the range.

The In-Place Text Editor opens and displays the formula in the cell.

5 Edit the formula, if necessary.

6 To save your changes and exit the editor, use one of the following methods:

■ Click OK on the toolbar.

■ Click in the drawing outside the editor.

■ Press CTRL+ENTER.

The cell displays the sum of the values in the range of cells. Empty cells and cells that do not resolve to a numeric value are ignored.

To average the values in a range of table cells

- 1 Select the table cell where you want to place the formula by clicking inside it.
- 2 Right-click. Click Insert Formula ► Average.
The following prompt is displayed:
Select first corner of table range:
- 3 Click inside the first cell in the range.
The following prompt is displayed:
Select second corner of table range:
- 4 Click inside the last cell in the range.
The In-Place Text Editor opens and displays the formula in the cell.
- 5 Edit the formula, if necessary.
- 6 To save your changes and exit the editor, use one of the following methods:
 - Click OK on the toolbar.
 - Click in the drawing outside the editor.
 - Press CTRL+ENTER.

The cell displays the average of the values in the range of cells. Empty cells and cells that do not resolve to a numeric value are ignored.

To count the cells in a table column or a table row

- 1 Select the table cell where you want to place the formula by clicking inside it.
- 2 Right-click. Click Insert Formula ► Count.
The following prompt is displayed:
Select first corner of table range:
- 3 Click inside the first cell in the range.
The following prompt is displayed:
Select second corner of table range:
- 4 Click inside the last cell in the range.
The In-Place Text Editor opens and displays the formula in the cell.
- 5 Edit the formula, if necessary.

6 To save your changes and exit the editor, use one of the following methods:

- Click OK on the toolbar.
- Click in the drawing outside the editor.
- Press CTRL+ENTER.

The cell displays the total number of cells in the range.

To use a cell from a different table in a formula

1 Select the table cell where you want to place the formula by clicking inside it.

2 Right-click. Click Insert Formula ► Cell.

The following prompt is displayed:

Select table cell:

3 Click inside the cell in the other table.

The In-Place Text Editor opens and displays the cell address.

4 Enter the rest of the formula.

5 To save your changes and exit the editor, use one of the following methods:

- Click OK on the toolbar.
- Click in the drawing outside the editor.
- Press CTRL+ENTER.

The cell displays the result of the calculation.

To manually enter a formula in a table cell

1 Double-click inside a table cell.

The In-place Text Editor opens.

2 Enter a formula (a function or an arithmetic expression), as in the following examples:

■ **=sum(a1:a25,b1)**. Sums the values in the first 25 rows of column A and the first row in column B.

■ **=average(a100:d100)**. Calculates the average of the values in the first 4 columns in row 100.

■ **=count(a1:m500)**. Displays the total number of cells in column A through column M in rows 1 through 100.

■ **=(a6+d6)/e1**. Adds the values in A6 and D6 and divides the total by the value in E1.

Use a colon to define a range of cells and a comma for individual cells. A formula must start with an equal sign (=) and can contain any of the following signs: plus (+), minus (-), times (*), divided by (/), exponent (^), and parentheses ().

- 3 To save your changes and exit the editor, use one of the following methods:

- Click OK on the toolbar.
- Click in the drawing outside the editor.
- Press CTRL+ENTER.

The cell displays the result of the calculation.

To turn off the display of column letters and row numbers for tables

- 1 On the command line, enter **tableindicator**.
- 2 At the Enter New Value prompt, enter **0**.

When TABLEINDICATOR is set to 1, the In-Place Text Editor displays column letters and row numbers when a table cell is selected.

To change the background color of column letters and row numbers for tables

- 1 Click a grid line to select a table.
- 2 Right-click. Click Table Color Indicator.
- 3 In the Select Color dialog box, select a color.
- 4 Click OK.

The text color, size, and style and the line color are controlled by the settings for column heads in the current table style.

To insert a Formula field in a table cell

- 1 Double-click inside a table cell.
- 2 In the In-Place Text Editor, right-click. Click Insert Field.
- 3 In the Field dialog box, Field Category list, select Objects.

- 4** In Field Names, select Formula.
- 5** To enter a formula, use any of the following methods one or more times:
 - Click Average, Sum, or Count. The Field dialog box closes temporarily. To specify a range, click inside the first and the last cell. The result is appended to the formula.
 - Click Cell. The Field dialog box closes temporarily. Select a cell in a table in the drawing. The cell address is appended to the formula.
- 6** (Optional) Select a format and a decimal separator.
- 7** Click OK.
- 8** To save your changes and exit the editor, use one of the following methods:
 - Click OK on the toolbar.
 - Click in the drawing outside the editor.
 - Press CTRL+ENTER.

The cell displays the result of the calculation.

28

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.

In this chapter

- Understand Basic Concepts of Dimensioning
- Use Dimension Styles
- Set the Scale for Dimensions
- Create Dimensions
- Modify Existing Dimensions
- Add Geometric Tolerances

Understand Basic Concepts of Dimensioning

You can create several types of dimensions, and you can control their appearance by setting up dimension styles or by editing individual dimensions.

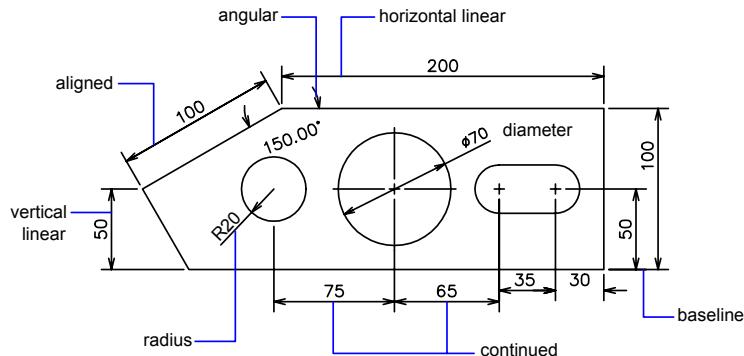
Overview of Dimensioning

Dimensioning is the process of adding measurement annotation to a drawing.

You can create dimensions for a variety of object types in many orientations. The basic types of dimensioning are

- Linear
- Radial (radius and diameter)
- Angular
- Ordinate
- Arc Length

Linear dimensions can be horizontal, vertical, aligned, rotated, baseline, or continued (chained). Some examples are shown in the illustration.



NOTE To simplify drawing organization and dimension scaling, it is recommended that you create dimensions on layouts rather than in model space.

To create a dimension

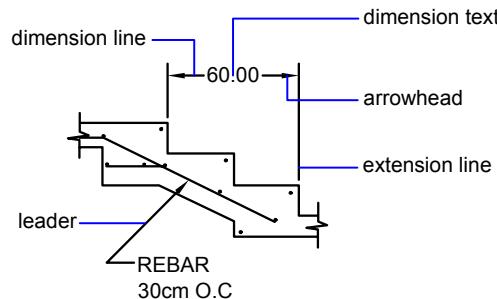
- 1 Create a layer designated for dimensions and make it the current layer.
- 2 Near the bottom-left corner of the application window, click a layout tab.
- 3 Click the Dimension menu and select a dimension command.
- 4 Follow the command prompts.

Command line: DIMLINEAR, DIMRADIUS, DIMANGULAR

Parts of a Dimension

Here is a list of the parts of a dimension along with their descriptions.

Dimensions have several distinct elements: dimension text, dimension lines, arrowheads, and extension lines.



Dimension text is a text string that usually indicates the measurement value. The text can also include prefixes, suffixes, and tolerances.

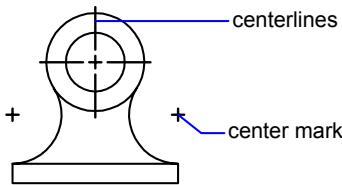
A *dimension line* indicates the direction and extent of a dimension. For angular dimensions, the dimension line is an arc.

Arrowheads, also called symbols of termination, are displayed at each end of the dimension line. You can specify different sizes and shapes for arrowheads or tick marks.

Extension lines, also called projection lines or witness lines, extend from the feature to the dimension line.

A *center mark* is a small cross that marks the center of a circle or arc.

Centerlines are broken lines that mark the center of a circle or arc.



Associative Dimensions

Dimensions can be associative, nonassociative, or exploded. Associative dimensions adjust to changes in the geometric objects that they measure.

Dimension associativity defines the relationship between geometric objects and the dimensions that give their distance and angles. There are three types of associativity between geometric objects and dimensions.

- **Associative dimensions**. Automatically adjust their locations, orientations, and measurement values when the geometric objects associated with them are modified. Dimensions in a layout may be associated to objects in model space. The DIMASSOC system variable is set to 2.
- **Nonassociative dimensions**. Selected and modified with the geometry they measure. Nonassociative dimensions do not change when the geometric objects they measure are modified. The dimension variable DIMASSOC is set to 1.
- **Exploded dimensions**. Contain a collection of separate objects rather than a single dimension object. The DIMASSOC system variable is set to 0.

You can determine whether a dimension is associative or nonassociative by selecting the dimension and doing one of the following:

- Use the Properties palette to display the properties of the dimension.
- Use the LIST command to display the properties of the dimension.

You can also use the Quick Select dialog box to filter the selection of associative or nonassociative dimensions. A dimension is considered associative even if only one end of the dimension is associated with a geometric object. The DIMREASSOCIATE command displays the associative and nonassociative elements of a dimension.

Special Situations and Limitations

You may need to use DIMREGEN to update associative dimensions after panning or zooming with a wheel mouse, after opening a drawing that was modified with an earlier release, or after opening a drawing with external references that have been modified.

Although associative dimensions support most object types that you would expect to dimension, they do not support the following:

- Hatchs
- Multiline objects
- 2D solids
- Objects with nonzero thickness

When selecting objects to dimension, make sure that the objects that you select don't include a directly overlapping object that does not support associative dimensioning such as a 2D solid.

Associativity is *not* maintained between a dimension and a block reference if the block is redefined.

Associativity is not maintained between a dimension and a 3D solid if the shape of the 3D solid is modified.

Dimensions created with QDIM are not associative but may be associated individually with DIMREASSOCIATE.

NOTE In releases prior to AutoCAD 2002, the definitions of associative and nonassociative dimensions were different and were controlled by the DIMASO system variable. The behavior of dimensions is now controlled by the DIMASSOC system variable.

For information about working with associative dimensions in combination with previous releases, see “Save Drawings to Previous Drawing File Formats”.

See also:

[“Change Dimension Associativity” on page 934](#)

To change the dimension associativity default

- 1 Click Tools menu ▶ Options.
- 2 In the Options dialog box, User Preferences tab, under Associative Dimensioning, select or clear Make New Dimensions Associative.

3 Do either or both of the following:

- Click **Apply** to record the current Options settings in the system registry.
- Click **OK** to record the current Options settings in the system registry and close the Options dialog box.

All subsequently created dimensions in the drawing use the new setting. Unlike most other option settings, dimension associativity is saved in the drawing file rather than in the system registry.

Command line: OPTIONS

Use Dimension Styles

You can control the appearance of dimensions by changing settings. For convenience and to help maintain dimensioning standards, you can store these settings in dimension styles.

Overview of Dimension Styles

A dimension style is a named collection of dimension settings that controls the appearance of dimensions, such as arrowhead style, text location, and lateral tolerances. You create dimension styles to specify the format of dimensions quickly, and to ensure that dimensions conform to industry or project standards.

- When you create a dimension, it uses the settings of the current dimension style
- If you change a setting in a dimension style, all dimensions in a drawing that use the style update automatically
- You can create *dimension substyles* that, for specified types of dimensions, deviate from the current dimension style
- If necessary, you can override a dimension style temporarily

To set the current dimension style

- On the Styles toolbar, in the Dimension Styles control, click the arrow and select a dimension style from the list.

Command line: DIMSTYLE

Compare Dimension Styles and Variables

You can view all the settings in a dimension style. Dimension styles used in externally referenced drawings are differentiated from those defined in your current drawing.

You can list the dimension styles in the current drawing. You can also list all dimensioning system variables and their current status or only the variables affected by a dimension style.

When you list the current status of all dimensioning system variables, any running overrides that apply to the current dimension style are listed. You can also list the differences between a named dimension style and the current dimension style.

Use Externally Referenced Dimension Styles

The program displays externally referenced dimension style names using the same syntax as for other externally dependent named objects. When you view externally referenced dimension styles using the Dimension Style Manager, the name of the xref displays in the Styles list as *Xref:drawing name* with each xref style appearing below the drawing name. For example, if the drawing file *baseplat.dwg* has a dimension style called FRACTIONAL-1, and you attach *baseplat.dwg* as an xref to a new drawing, then the xref dimension style is displayed in the Styles list of the Dimension Style Manager as *Xref:baseplat.dwg*, and FRACTIONAL-1 appears under the drawing name.

Externally referenced dimension styles can be examined, but they cannot be modified or made current. You can use an externally referenced dimension style as a template for creating a new dimension style in your current drawing.

To list all dimension settings for the current dimension style

- 1 Click Dimension menu ► Style.
- 2 In the Dimension Style Manager, select the style from the Styles list.
- 3 Click Compare.

The dimensioning system variables, their current settings, and a brief description are listed. Overrides are included.

Dimension or Styles toolbar



Command line: DIMSTYLE

To list settings for an existing dimension style

- 1 Click Dimension menu ▶ Style.
- 2 In the Dimension Style Manager, enter a dimension style name, or select a dimension whose dimension style you want to examine.
- 3 Click Compare.

Affected variables, their settings, and a brief description of each are listed. Overrides are not included.

Dimension or Styles toolbar



Command line: DIMSTYLE

To list dimension styles in the current drawing

- 1 Click Dimension menu ▶ Style.
- 2 In the Dimension Style Manager, under List, select All Styles or Style in Use.

Dimension or Styles toolbar



Command line: DIMSTYLE

To compare dimension styles

- 1 Click Dimension menu ▶ Style.
- 2 In the Dimension Style Manager, select the style to compare from the Styles list.
- 3 Click Compare.

The dimension style is compared to the current dimension style.

Dimension or Styles toolbar



Command line: DIMSTYLE

Control Dimension Geometry

You can control the appearance of dimension lines, extension lines, arrowheads, and center marks.

Control Dimension Lines

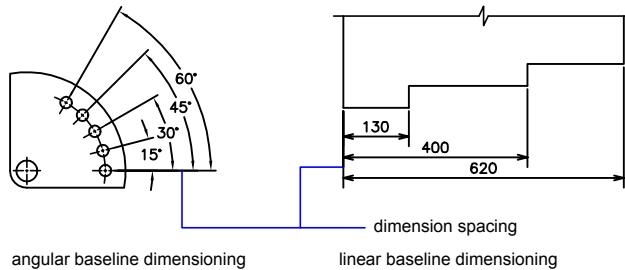
You can control dimension line properties including color, linewidth, and spacing.

You can control several aspects of a dimension line. You can

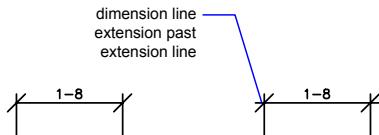
- Specify color and linewidth for visual effect and plotting
- Suppress the dimension line or, if the dimension line is broken by text, one or both halves



- Control the spacing between successive dimension lines in baseline dimensions



- Control the distance by which the dimension line extends beyond the extension lines for architectural tick (oblique stroke) arrowheads



To modify the display of dimension lines

- 1 Click Dimension menu ► Style.
- 2 In the Dimension Style Manager, select the style you want to change. Click Modify.
- 3 In the Modify Dimension Style dialog box, Lines tab, change the settings under Dimension Lines as needed.
- 4 Click OK.
- 5 Click Close to exit the Dimension Style Manager.

Dimension or Styles toolbar



Command line: DIMSTYLE

Control Extension Lines

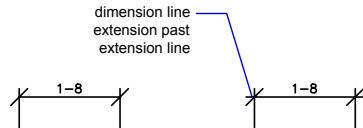
You can control extension line properties including color, linewidth, overshoot, and offset length.

You can control several aspects of the extension lines. You can

- Specify color and linewidth for visual effect and plotting
- Suppress one or both extension lines if they are unnecessary, or if there is not enough space



- Specify how far beyond the dimension line that the extension line extends (overshoot)



- Control the extension origin offset, the distance between the extension line origin, and the start of the extension line



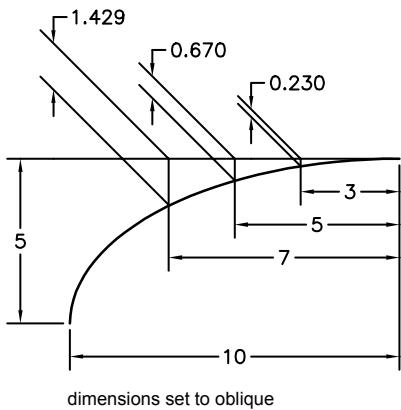
- Specify a fixed length for extension lines, as measured from the dimension line toward the extension line origin



- Specify a noncontinuous linetype, typically used for centerlines

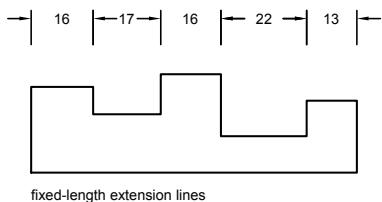


- Modify the angle of the extension lines of a selected dimension to make them oblique



Fixed-Length Extension Lines

With the Dimension Style Manager, on the Lines tab, you can specify a dimension style that sets the total length of extension lines starting from the dimension line toward the dimension origin point.



The extension-line offset distance from the origin will never be less than the value specified by the DIMEXO system variable.

See also:

["Create Dimensions with Oblique Extension Lines" on page 914](#)

To modify the display of extension lines

- 1 Click Dimension menu ▶ Style.
- 2 In the Dimension Style Manager, select the style you want to change. Click Modify.

- 3 In the Modify Dimension Style dialog box, Lines tab, under Extension Lines, change the settings as needed.
- 4 Click OK.
- 5 Click Close to exit the Dimension Style Manager.

Dimension or Styles toolbar



Command line: DIMSTYLE

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 888](#)

To choose an arrowhead

- 1 Click Dimension menu ▶ Style.
- 2 In the Dimension Style Manager, select the style you want to change. Click Modify.
- 3 In the Modify Dimension Style dialog box, Symbols and Arrows tab, under Arrowheads, select the arrowhead type for the first end of the dimension line.

The second arrowhead is automatically set to the same type.

- 4 To set the second end of the dimension line to a different arrowhead type, select an arrowhead type from the Second list.
- 5 In the Size box, enter a size for the arrowhead.
- 6 Click OK.
- 7 Click Close to exit the Dimension Style Manager.

Dimension or Styles toolbar



Command line: DIMSTYLE

To flip the direction of an arrowhead

- 1 At the Command prompt, select a single dimension object near the arrowhead that you want to flip.
- 2 Right-click. Click Flip Arrow.

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 461.

Arrowhead sizing relies on the overall dimension scale factor. When you create a dimension, the block is inserted where the arrowheads would normally go. The object's X and Y scale factors are set to *arrowhead size overall scale*. The dimension line is trimmed by *text gap x overall scale* units at each end. To trim the dimension line, the rightmost block is inserted with a zero rotation angle for horizontal dimensioning. The leftmost block is rotated 180 degrees about its insertion point.

If you use paper-space scaling, the scale factor is computed before applying it to the arrowhead size value.

To use your own arrowhead symbol

- 1 Click Dimension menu ► Style.

- 2 In the Dimension Style Manager, select the style you want to change. Click Modify.
- 3 In the Dimension Style Manager, Symbols and Arrows tab, under Arrowheads, select User Arrow from the First arrowhead list.
- 4 In the Select Custom Arrow Block dialog box, enter the name of your block. Click OK.
- 5 To choose a different custom arrowhead for the second arrowhead, repeat steps 3 and 4, choosing User Arrow from the Second arrowhead list (optional).
- 6 Click OK.
- 7 Click Close to exit the Dimension Style Manager.

Dimension or Styles toolbar



Command line: DIMSTYLE

Control Dimension Text

You can control the placement of dimension text, arrowheads, and leader lines relative to the dimension and extension lines.

Fit Dimension Text Within Extension Lines

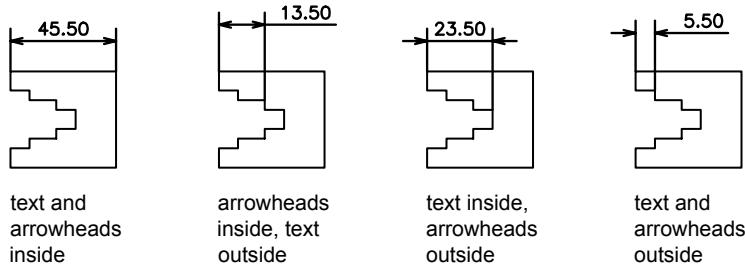
Dimension text and arrowheads usually appear between the extension lines when there is enough space. You can specify how these elements are placed when space is limited.

Many factors, such as the size of extension line spacing and arrowhead size, influence how dimension text and arrowheads fit within the extension lines. In general, the best fit, given the available space, is applied. If possible, both text and arrowheads are accommodated between the extension lines, no matter what fit option you choose.

When creating new dimensions, you can choose to place text by entering a coordinate or using the pointing device; this is known as user-defined text placement. Alternatively, the program can compute the text position for you. The options for automatic fitting of text and arrowheads are listed in the Dimension Style Manager, Fit tab. For example, you can specify that text and arrowheads be kept together. In this case, if there is not room for both between the extension lines, they are both placed outside. You can specify that if there

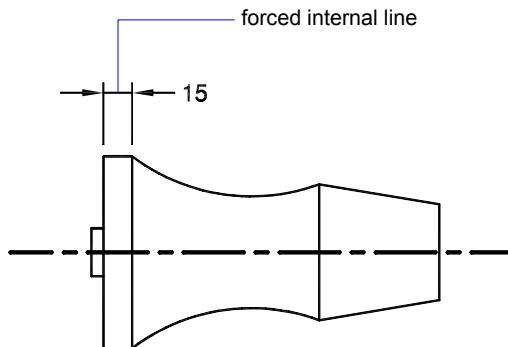
is room for only text or arrowheads, then either text only or arrowheads only are placed between the extension lines.

The following illustrations show how the program applies a "best fit" for arrowheads and text.



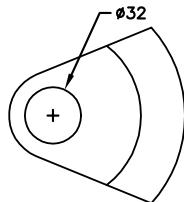
If there is no room for text between the extension lines, you can have a leader line created automatically. This is useful in cases where text outside the extension lines would interfere with other geometry, for example, in continued dimensions. Whether text is drawn to the right or the left of the leader is controlled by the horizontal justification setting on the Text tab of the Modify/New Dimension Style dialog box. Also, you can fit text and arrowheads by changing their size.

Even if the arrowheads are outside the extension lines, you can have a line drawn between the extension lines. This is called forcing an internal line and is illustrated as follows.

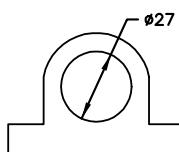


Fit Diameter Dimension Text

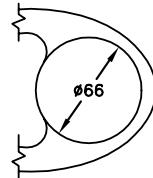
You can draw several different diameter dimensions depending on text placement, horizontal settings on the Text tab, and whether you select the Draw Dim Line Between Ext Lines option on the Fit tab.



default horizontal
placement-text outside
circle, center mark, no
forced interior line



user-defined horizontal
placement-dimension line
and arrows, no center
mark, forced interior line



Text and Arrows inside
with Inside Horizontal
option selected

To place text within extension lines

- 1 Click Dimension menu ► Style.
- 2 In the Dimension Style Manager, select the style you want to change. Click Modify.
- 3 In the Modify Dimension Style dialog box, Fit tab, under Fit Options, select an option.
- 4 Click OK.
- 5 Click Close to exit the Dimension Style Manager.

If there is enough room, text is fit between extension lines.

Dimension or Styles toolbar



Command line: DIMSTYLE

To force an internal line and choose a fit option

- 1 Click Dimension menu ► Style.
- 2 In the Dimension Style Manager, select the style you want to change. Click Modify.

- 3 In the Modify Dimension Style dialog box, Fit tab, Under Fine Tuning, select Always Draw Dim Line Between Ext Lines.
- 4 Under Fit Options, select an option.
- 5 Click OK.
- 6 Click Close to exit the Dimension Style Manager.

Dimension or Styles toolbar

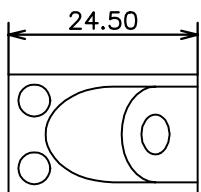


Command line: DIMSTYLE

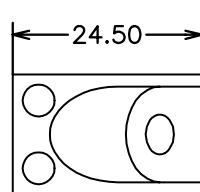
Control the Location of Dimension Text

You can locate dimension text manually and specify its alignment and orientation.

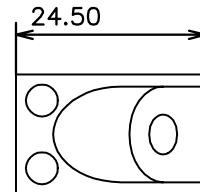
The program comes with several justification settings that facilitate compliance with international standards, or you can choose your own location for the text.



text centered above the dimension line



text centered horizontally and vertically

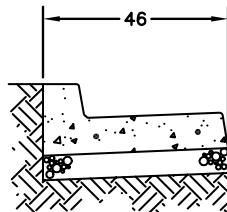


text left-justified above the dimension line

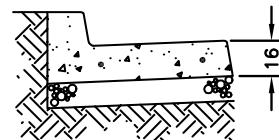
Many of the settings are interdependent. Example images in the Dimension Style Manager are updated dynamically to illustrate how text appears as you change the settings.

Align Dimension Text

Whether text is inside or outside the extension lines, you can choose whether it is aligned with the dimension line or remains horizontal. The following examples show two combinations of these options.



text inside extension lines
oriented horizontally

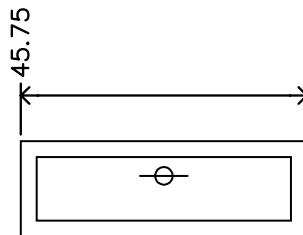


text outside extension lines
aligned with dimension line

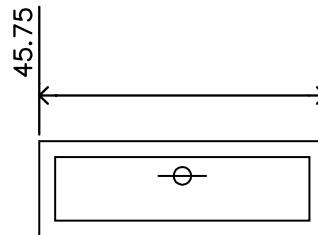
The default alignment is horizontal dimension text, even for vertical dimensions.

Position Dimension Text Horizontally

The position of the text along the dimension line in relation to the extension lines is referred to as text placement. To place text yourself when you create a dimension, use the Place Text Manually When Dimensioning option on the Fit tab of the Modify/New Dimension Style dialog box. Use the text placement options to automatically place text at the center of the dimension line, at either extension line, or over either extension line.

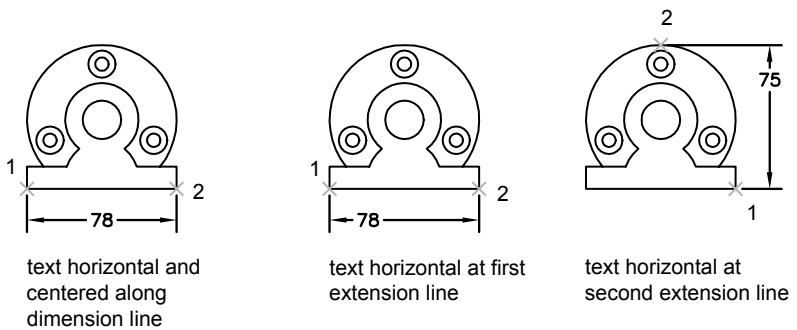


horizontal justification over
extension line one, vertical
justification of text, Centered over
extension line



horizontal justification over
extension line one, vertical
justification of text, Above
extension line

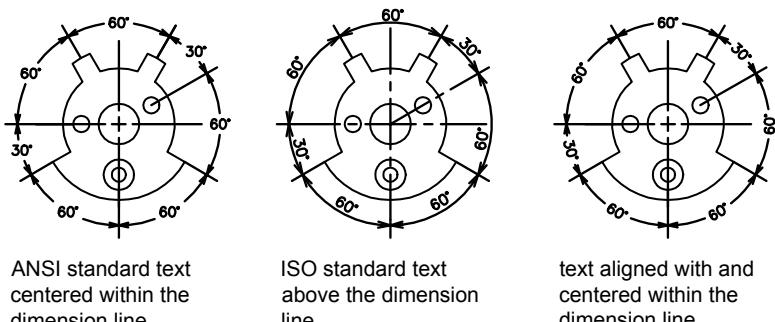
First and *second* extension lines are defined by the order in which you specified the extension line origins when you created the dimension. For angular dimensions, the second extension line is counterclockwise from the first. In the following illustrations, 1 is the first extension line origin and 2 the second.



If you place text manually, you can place the dimension text anywhere along the dimension line, inside or outside the extension lines, as you create the dimension. This option provides flexibility and is especially useful when space is limited. However, the horizontal alignment options provide better accuracy and consistency between dimensions.

Position Dimension Text Vertically

The position of the text relative to the dimension line is referred to as vertical text placement. Text can be placed above or below or centered within the dimension line. In the ANSI standards, centered text usually splits the dimension line. In the ISO standards, it is usually above or outside the dimension line. For example, ISO standards permit angular dimension text to appear in any of the ways shown.



Other settings, such as Text Alignment, affect the vertical alignment of text. For example, if Horizontal Alignment is selected, text inside the extension lines and centered within the dimension line is horizontal, as shown in the leftmost illustration above. The text is horizontal even if the dimension line is not itself horizontal.

To align text with the dimension line

- 1 Click Dimension menu ► Style.
- 2 In the Dimension Style Manager, select the style you want to change. Click Modify.
- 3 In the Modify Dimension Style dialog box, Text tab, under Text Alignment, select Aligned with Dimension Line.
- 4 Click OK.
- 5 Click Close to exit the Dimension Style Manager.

Dimension or Styles toolbar



Command line: DIMSTYLE

To place text at the second extension line

- 1 Click Dimension menu ► Style.
- 2 In the Dimension Style Manager, select the style you want to change. Click Modify.
- 3 In the Modify Dimension Style dialog box, Text tab, under Text Placement, select Over Ext Line 2 from the Horizontal list box.
The example area reflects your selection.
- 4 Click OK.
- 5 Click Close to exit the Dimension Style Manager.

Dimension or Styles toolbar



Command line: DIMSTYLE

To place dimension text manually

- 1 Click Dimension menu ► Style.
- 2 In the Dimension Style Manager, select the style you want to change. Click Modify.
- 3 In the Modify Dimension Style dialog box, Fit tab, under Fine Tuning, select Place Text Manually When Dimensioning.
- 4 Click OK.
- 5 Click Close to exit the Dimension Style Manager.

As you create dimensions, you can move the text along the dimension line. Use the pointing device or enter coordinates to specify the dimension line and text locations.

Dimension or Styles toolbar



Command line: DIMSTYLE

To place text above the dimension line

- 1 Click Dimension menu ► Style.
- 2 In the Dimension Style Manager, select the style you want to change. Click Modify.
- 3 In the Modify Dimension Style dialog box, Text tab, under Text Placement, select Above from the Vertical list box.
The example area reflects your selection.
- 4 Click OK.
- 5 Click Close to exit the Dimension Style Manager.

Dimension or Styles toolbar



Command line: DIMSTYLE

Control the Appearance of Dimension Text

You can include prefixes, suffixes, and user-supplied text in dimensions. You can also control the text style and formatting used in dimension text.

The program supports a mixture of user-supplied text, prefixes and suffixes supplied by the dimension style, and generated measurements. For example, you could add a diameter symbol as a prefix to a measurement or add the abbreviation for a unit, such as mm, as a suffix. Text in this context refers to all dimension text, prefixes and suffixes, primary and alternate units, and lateral tolerances. Geometric tolerances are controlled independently.

Dimension text is treated as a single string of text, which you create and format using your text editor.

Control the Text Style in Dimensions

The appearance of dimension text is governed by the text style selected in the Dimension Style Manager, Text tab. You can choose a text style while creating a dimension style and specify a text color and a height independent of the current text style's height setting. You can also specify the gap between base dimension text and the box that surrounds it.

The text styles used for dimensions are the same text styles used by all text created in your drawing.

For more information, see “Work with Text Styles” on page 831.

Supply User Text to Dimensions

In addition to the prefixes and suffixes specified for primary and alternate units, you can supply your own text as you create a dimension. Because the prefix, suffix, and user-supplied text form a single text string, you can represent tolerance stacks and apply changes to font, text size, and other characteristics using the text editor.

To add user text above and below the dimension line, use the separator symbol \X. Text that precedes this symbol is aligned with and above the dimension line. Text that follows the \X symbol is aligned with and below the dimension line. The space between the dimension line and the text is determined by the value you enter under Gap in the Annotation dialog box.

Example: User Text in Dimensions

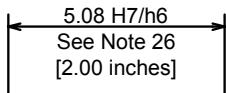
In this example, the primary dimension measurement is 5.08, and the alternate dimension measurement is 2.00. The primary units have the suffix *H7/h6*, and the alternate units have the suffix *inches*.

At the text prompt, while creating the dimension, you enter the following format string:

<> H7/h6\XSee Note 26\P[]

The angle brackets represent the primary units, and the square brackets represent the alternate units. The \X separates text above the dimension line from text below the dimension line. The \P is a paragraph break.

The resulting text appears as follows:



To control the text style in dimensions

- 1 Click Dimension menu ► Dimension Style.
- 2 In the Dimension Style Manager, select the style you want to change. Click Modify.
- 3 In the Modify Dimension Style dialog box, Text tab, under Text Appearance, select a text style.
- 4 If the current text style does not have a fixed height, enter the height of dimension text in the Text Height box.
- 5 Under Tolerances, enter a height for tolerance values in the Scaling for Height box.
- 6 In the Offset from Dim Line box, enter a value for the gap around base dimension text.
- 7 Select a color from the Text Color box.
- 8 Click OK
- 9 Click Close to exit the Dimension Style Manager.

Dimension or Styles toolbar



Command line: DIMSTYLE

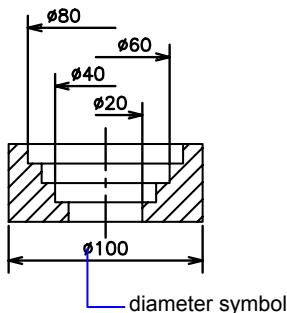
Control Dimension Values

The numeric values displayed in dimensions can appear in several formats. You can also control how numeric distances are represented.

Control the Display of Dimension Units

The numeric values of dimensions can be displayed as a single measurement or in two measurement systems. In either case, you can control details of how the numeric values are presented.

The settings for primary units control the display of the dimension values, including the unit format, the numeric precision, and the decimal separator style. For example, you can enter the diameter symbol as a prefix, as shown in the illustration. Any prefix you specify replaces the prefixes normally used for diameter and radius dimensions (Diameter (unicode 2205 and R, respectively)).



These settings are available on the Primary Units tab of the Dimension Style Manager.

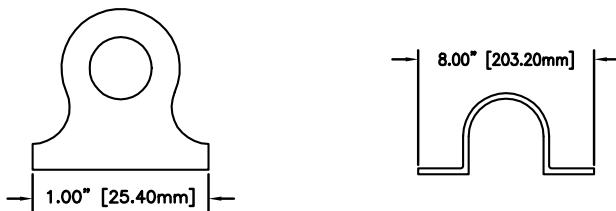
Control the Display of Alternate Units

You can create dimensions in two systems of measurement simultaneously. A common use of this feature is to add feet and inches dimensions to drawings created using metric units. The alternate units appear in square brackets ([]) in the dimension text. Alternate units cannot be applied to angular dimensions.

If alternate-units dimensioning is on when you edit a linear dimension, the measurement is multiplied by an alternate scale value that you specify. This value represents the number of alternate units per current unit of measurement. The default value for imperial units is 25.4, which is the number of millimeters

per inch. The default value for metric units is about 0.0394, which is the number of inches per millimeter. The number of decimal places is specified by the precision value for alternate units.

For example, for imperial units, if the alternate scale setting is the default value, 25.4, and the alternate precision is 0.00, the dimension might look like the following figure.



To add and format primary units

- 1 Click Dimension menu ► Style.
- 2 In the Dimension Style Manager, select the style you want to change. Click Modify.
- 3 In the Modify Dimension Style dialog box, Primary Units tab, under Linear or Angular Dimensions, select a unit format and precision value for the primary units.
- 4 Under Linear Dimensions, enter any prefix and suffix for the displayed dimension.
- 5 Click OK.
- 6 Click Close to exit the Dimension Style Manager.

Dimension or Styles toolbar



Command line: DIMSTYLE

To add and format alternate units

- 1 Click Dimension menu ► Style.

- 2** In the Dimension Style Manager, select the style you want to change. Click Modify.
- 3** In the Modify Dimension Style dialog box, Alternate Units tab, select Display Alternate Units.
- 4** Under Alternate Units
 - Select a unit format from the list.
 - Select a precision value for the alternate units.
 - Enter any prefix and suffix for the displayed dimension, including a space character if you want a gap between the dimension and the prefix or suffix.
- 5** Click OK.
- 6** Click Close to exit the Dimension Style Manager.

Dimension or Styles toolbar

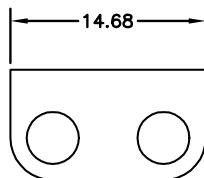


Command line: DIMSTYLE

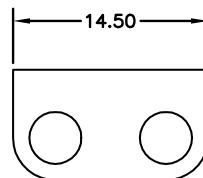
Round Off Dimension Values

You can round off the numeric values in dimensions and lateral tolerances.

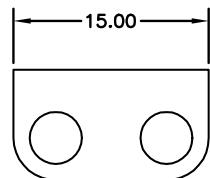
You can round off all dimension values except those for angular dimensions. For example, if you specify a round-off value of 0.25, all distances are rounded to the nearest 0.25 unit. The number of digits displayed after the decimal point depends on the precision set for primary and alternate units and lateral tolerance values.



round-off value set to default (0)



round-off value set to default .25



round-off value set to default 1

To round off dimension values

- 1 Click Dimension menu ► Style.
- 2 In the Dimension Style Manager, select the style you want to change. Click Modify.
- 3 In Modify Dimension Style dialog box, Primary Units tab, under Linear Dimensions, enter the round-off value.
- 4 Click OK.
- 5 Click Close to exit the Dimension Style Manager.

Dimension or Styles toolbar



Command line: DIMSTYLE

SUPPRESS ZEROS IN DIMENSIONS

You can suppress leading and trailing zeros in the numeric portion of dimension text.

If you suppress leading zeros in decimal dimensions, 0.500 becomes .500. If you suppress trailing zeros, 0.500 becomes 0.5. You can suppress *both* leading and trailing zeros so that 0.5000 becomes .5 and 0.0000 becomes 0.

The table shows the effect of selecting each option and provides examples of the architectural units style. If feet are included with a fractional inch, the number of inches is indicated as zero, no matter which option you select. Thus, the dimension 4'-3/4" becomes 4'-0 3/4".

Zero suppression for feet and inches

Option	Effect	Examples			
No options selected	Includes zero feet and zero inches	0'-0 1/2"	0'-6"	1'-0"	1'-0 3/4"
0 Inches selected	Suppresses zero inches (includes zero feet)	0'-0 1/2"	0'-6"	1'	1'-0 3/4"
0 Feet selected	Suppresses zero feet (includes zero inches)	1/2"	6"	1'-0"	1'-0 3/4"
0 Feet and 0 Inches selected	Suppresses zero feet and zero inches	1/2"	6"	1'	1'-0 3/4"

To suppress zeros in dimension values

- 1 Click Dimension menu ► Style.
- 2 In the Dimension Style Manager, select the style you want to change. Click Modify.
- 3 In the Modify Dimension Style dialog box, Primary Units tab or Alternate Units tab, under Zero Suppression, select from the following:
 - **Leading.** Suppresses leading zeros in decimal values.
 - **Trailing.** Suppresses trailing zeros in decimal values.
 - **0 Feet.** Suppresses display of 0 feet in feet and inches values.
 - **0 Inches.** Suppresses display of 0 inches in feet and inches values.
- 4 Click OK.
- 5 Click Close to exit the Dimension Style Manager.

Dimension or Styles toolbar



Command line: DIMSTYLE

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.

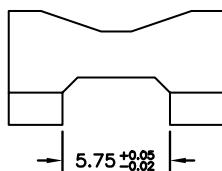
A lateral tolerance specifies the amount by which a dimension can vary. By specifying tolerances in manufacturing, you can control the degree of accuracy needed for a feature. A feature is some aspect of a part, such as a point, line, axis, or surface.

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.

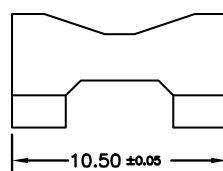
Lateral tolerances can be specified from theoretically exact measurements. These are called basic dimensions and have a box drawn around them.

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 \pm sign and they are known as symmetrical. Otherwise, the plus value goes above the minus value.

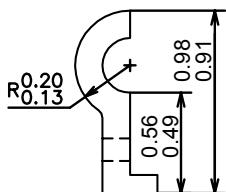


deviation tolerance



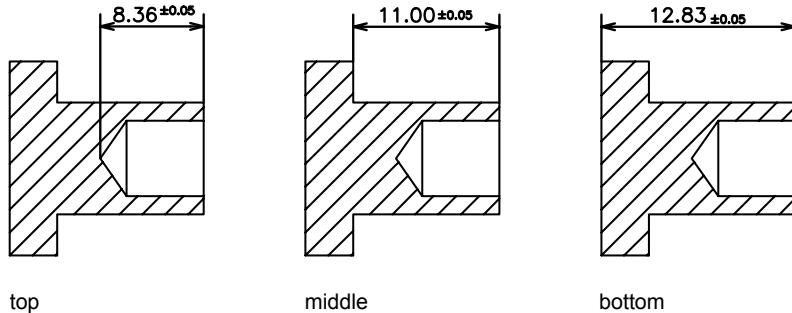
symmetrical deviation tolerance

If the tolerances are applied as limits, the program uses the plus and minus values you supply to calculate a maximum and minimum value. These values replace the dimension value. If you specify limits, the upper limit goes above the lower.



Format Lateral Tolerances

You can control the vertical placement of tolerance values relative to the main dimension text. Tolerances can align with the top, middle, or bottom of the dimension text.



You can also control zero suppression as you can with the primary and alternate units. Suppressing zeros in lateral tolerances has the same effect as suppressing them in the primary and alternate units. If you suppress leading zeros, .5 becomes .5, and if you suppress trailing zeros, 0.5000 becomes 0.5.

See also:

["Add Geometric Tolerances" on page 936](#)

To specify methods for lateral tolerances

- 1 Click Format menu ► Dimension Style.
- 2 In the Dimension Style Manager, select the style you want to change. Click Modify.
- 3 In the Modify Dimension Style dialog box, Tolerances tab, under Tolerance Format, select a method from the Method list, and then do one of the following:
 - If you select Limits, enter upper and lower tolerance deviation in the Upper Value and Lower Value boxes.
 - If you select Symmetrical tolerances, Lower Value is not available, because you need only one tolerance value.
 - If you select Basic, enter a value in Offset from Dim Line (on the Text tab) to represent the gap between the text and its enclosing box.
- 4 Click OK.
- 5 Click Close to exit the Dimension Style Manager.

Dimension or Styles toolbar



Command line: DIMSTYLE

To align and suppress zeros in tolerance values

- 1 Click Dimension menu ► Style.
- 2 In the Dimension Style Manager, select the style you want to change. Click Modify.
- 3 In the Modify Dimension Style dialog box, Tolerances tab, under Tolerance Format, select the alignment from the Vertical Position list.
- 4 To suppress zeros in primary or alternate units, under Zero Suppression, select Leading to suppress leading zeros. Select Trailing to suppress trailing zeros.
- 5 Click OK.
- 6 Click Close to exit the Dimension Style Manager.

Dimension or Styles toolbar



Command line: DIMSTYLE

Set the Scale for Dimensions

You can specify the size of dimensions in your drawing. How you set dimension size depends on the method you use to lay out and plot drawings.

Dimension scale affects the size of the dimension geometry relative to the objects in the drawing. Dimension scale affects sizes, such as text height and arrowhead size, and offsets, such as the extension line origin offset. You should set these sizes and offsets to values that represent their actual plotted size. Dimension scale does *not* apply the overall scale factor to tolerances or measured lengths, coordinates, or angles.

Setting dimension scale depends on how you lay out your drawing. There are three methods used to create dimensions in a drawing layout:

- **Dimension in model space for plotting in model space.** This is the traditional method used with single-view drawings. To create dimensions that are scaled correctly for plotting, set the DIMSCALE system variable to

the inverse of the intended plot scale. For example, if the plot scale is 1/4, set DIMSCALE to 4.

■ **Dimension in model space for plotting in paper space.** This was the preferred method for complex, multiple-view drawings prior to AutoCAD 2002. Use this method when the dimensions in a drawing need to be referenced by other drawings (xrefs) or when creating isometric dimensions in 3D isometric views. To prevent the dimensions in one layout viewport from being displayed in other layout viewports, create a dimensioning layer for each layout viewport that is frozen in all other layout viewports. To create dimensions that are scaled automatically for display in a paper space layout, set the DIMSCALE system variable to 0.

■ **Dimension in layouts.** This is the simplest dimensioning method. Dimensions are created in paper space by selecting model space objects or by specifying object snap locations on model space objects. By default, associativity between paper space dimensions and model space objects is maintained. No additional scaling is required for dimensions created in a paper space layout: DIMLFAC and DIMSCALE do not need to be changed from their default value of 1.0000.

NOTE When you dimension model space objects in paper space using associative dimensions, dimension values for the display scale of each viewport are automatically adjusted. This adjustment is combined with the current setting for DIMLFAC and is reported by the LIST command as a dimension style override. For nonassociative dimensions, you must set DIMLFAC manually.

See also:

["Draw, Scale, and Annotate in Model Space" on page 204](#)

["Scale Views in Layout Viewports" on page 223](#)

To set the overall dimension scale

- 1 Click Dimension menu ► Style.
- 2 In the Dimension Style Manager, select the style you want to change. Click Modify.
- 3 In Modify Dimension Style dialog box, Fit tab, under Scale for Dimension Features, enter a value for the overall scale.
- 4 Click OK.
- 5 Click Close to exit the Dimension Style Manager.

Dimension or Styles toolbar



Command line: DIMSTYLE

To set the dimension scale for model space dimensions in layouts

- 1 Click Dimension menu ► Style.
- 2 In the Dimension Style Manager, select the style you want to change. Click Modify.
- 3 In Modify Dimension Style dialog box, Fit tab, under Scale for Dimension Features, select Scale Dimension to Layout (Paperspace).
- 4 Click OK.
- 5 Click Close to exit the Dimension Style Manager.

Dimension or Styles toolbar



Command line: DIMSTYLE

To set dimension scale for creating dimensions in a layout

- 1 Click a layout tab to switch to paper space.
 - 2 To create dimensions in paper space with the correct model space dimension values, use object snap modes to snap to points in model space from paper space or select the objects directly.
- The DIMLFAC system variable can be changed if you need to convert the linear dimension values between the imperial and metric measurement systems.

Create Dimensions

You can create all of the standard types of dimensions.

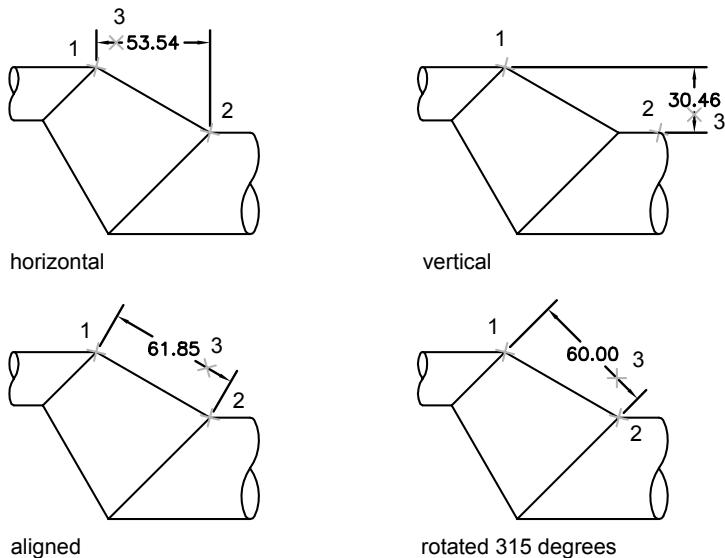
Create Linear Dimensions

You can create linear dimensions with horizontal, vertical, and aligned dimension lines. These linear dimensions can also be stacked, or they can be created end to end.

Overview of Creating Linear Dimensions

Linear dimensions can be horizontal, vertical, or aligned. With aligned dimensions, the dimension line is parallel to the line (imaginary or real) between the extension line origins. Baseline (or parallel) and continued (or chain) dimensions are series of consecutive dimensions that are based on a linear dimension.

In all four illustrations, the extension line origins are designated explicitly at 1 and 2, respectively. The dimension line location is specified at 3.



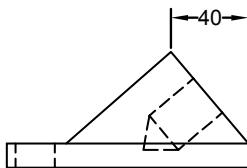
As you create linear dimensions, you can modify the content of the text, the angle of the text, or the angle of the dimension line.

Create Horizontal and Vertical Dimensions

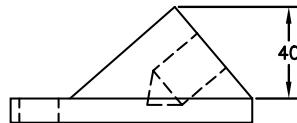
You can create dimensions using only the horizontal or vertical components of the locations or objects that you specify.

The program automatically applies a horizontal or vertical dimension according to the extension line origins that you specify or the location where you select an object; however, you can override this as you create the dimension by specifying that a dimension be horizontal or vertical. For example, in the

following illustration, a horizontal dimension is drawn by default unless you specify a vertical one.



horizontal dimension created
by default



vertical dimension specified

To create a horizontal or vertical dimension

- 1 Click Dimension menu ► Linear.
- 2 Press ENTER to select the object to dimension, or specify the first and second extension line origins.
- 3 Before specifying the dimension line location, you can override the dimension direction and edit the text, the text angle, or the dimension line angle:
 - To rotate the extension lines, enter **r** (Rotated). Then enter the dimension line angle.
 - To edit the text, enter **m** (multiline text). In the In-place Text Editor, revise the text. Click OK.
Editing within or overwriting the brackets (<>) changes or removes the dimension value calculated by the program. Adding text before or after the brackets appends text before or after the dimension value.
 - To rotate the text, enter **a** (Angle). Then enter the text angle.
- 4 Specify the dimension line location.

Dimension toolbar

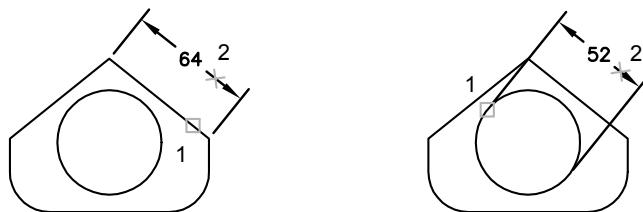


Command line: DIMLINEAR

Create Aligned Dimensions

You can create dimensions that are parallel to the locations or objects that you specify.

In aligned dimensions, the dimension line is parallel to the extension line origins. The illustration shows two examples of aligned dimensioning. The object is selected (1), and the location of the aligned dimension is specified (2). The extension lines are drawn automatically.



To create an aligned dimension

- 1 Click Dimension menu ▶ Aligned.
- 2 Press ENTER to select the object to dimension, or specify the first and second extension line origins.
- 3 Before specifying the dimension line location, you can edit the text or change the text angle. Editing within or overwriting the brackets (<>) changes or removes the dimension value calculated by the program. Adding text before or after the brackets appends text before or after the dimension value.
 - To edit the text using multiline text, enter **m** (multiline text). In the In-place Text Editor, revise the text. Click OK.
 - To edit the text using single-line text, enter **t** (Text). Revise the text on the command line and press ENTER .
 - To rotate the text, enter **a** (Angle). Then enter the text angle.
- 4 Specify the dimension line location.

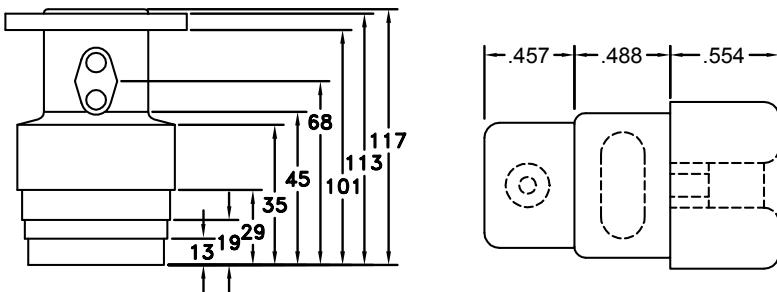
Dimension toolbar



Command line: DIMALIGNED

Create Baseline and Continued Dimensions

Baseline dimensions are multiple dimensions measured from the same baseline. Continued dimensions are multiple dimensions placed end to end. You must create a linear, aligned, or angular dimension before you create baseline or continued dimensions. You create baseline dimensions incrementally from the most recently created dimension in the current session.



Both baseline and continued dimensions are measured from the previous extension line unless you specify another point as the point of origin.

To create a baseline linear dimension

- 1 Click Dimension menu ► Baseline.

By default, the origin of the last linear dimension created is used as the first extension line for the new baseline dimension. You are prompted for the second dimension line.

- 2 Use an object snap to select the second extension line origin, or press ENTER to select any dimension as the base dimension.

The program automatically places the second dimension line at the distance specified by the Baseline Spacing option in the Dimension Style Manager, Lines tab.

- 3 Use an object snap to specify the next extension line origin.
- 4 Continue to select extension line origins as required.

5 Press ENTER twice to end the command.

Dimension toolbar



Command line: DIMBASELINE

To create a continued linear dimension

1 Click Dimension menu ► Continue.

The program uses the origin of the second extension line of the existing dimension as the first extension line origin.

2 Use object snaps to specify additional extension line origins.

3 Press ENTER twice to end the command.

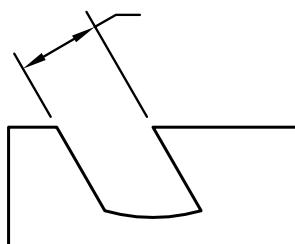
Dimension toolbar



Command line: DIMCONTINUE

Create Rotated Dimensions

In rotated dimensions, the dimension line is placed at an angle to the extension line origins. The illustration shows an example of a rotated dimension. In the example, the angle specified for dimension rotation is equal to the angle of the slot.



To create a rotated dimension

- 1 Click Dimension menu ► Linear.
- 2 Press ENTER to select the object to dimension or specify the first and second extension line origins.
- 3 To rotate the dimension line, enter **r** (Rotated). Then enter the dimension line angle.
- 4 Specify the dimension line location.

Dimension toolbar



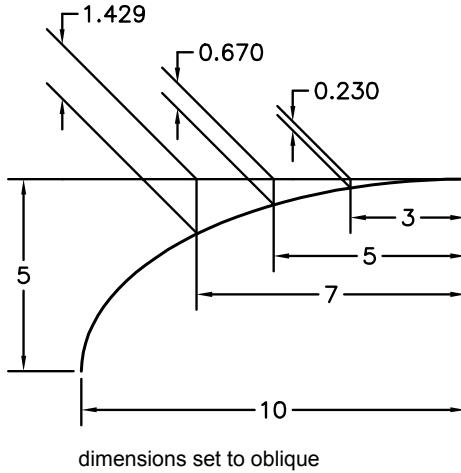
Command line: DIMLINEAR

Create Dimensions with Oblique Extension Lines

You can create dimensions with extension lines that are not perpendicular to their dimension lines.

Extension lines are created perpendicular to the dimension line. However, if the extension lines conflict with other objects in a drawing, you can change their angle after the dimension has been drawn.

New dimensions are *not* affected when you make an existing dimension oblique.



To make extension lines oblique

- 1 Click Dimension menu ► Oblique.
- 2 Select the dimension.
- 3 Enter a value for the angle of obliqueness, or specify two points.

Dimension toolbar



Command line: DIMEDIT

Create Radial Dimensions

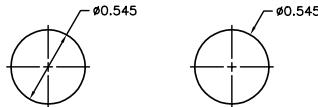
Radial dimensions measure the radii and diameters of arcs and circles with optional centerlines or a center mark.

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.



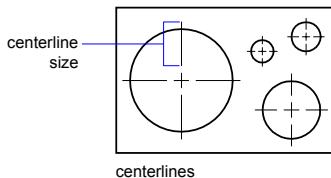
- **DIMDIAMETER** measures the diameter of an arc or circle, and displays the dimension text with the diameter symbol in front of it.



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 Centerlines and Center Marks

Depending on your dimension style settings, center marks and lines generate automatically for diameter and radius dimensions. They are created only if the dimension line is placed outside the circle or arc. You can create centerlines and center marks directly with the **DIMCENTER** command.



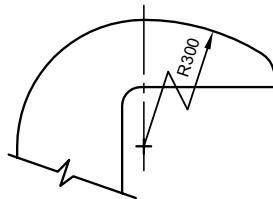
You can control the size and visibility of centerlines and center marks on the Modify Dimension Style dialog box, Symbols and Arrows tab, under Center Marks. You can also access this setting with the **DIMCEN** system variable.

The size of the centerline is the length of the centerline segment that extends outside the circle or arc. It is also the size of the gap between the center mark and the start of the centerline.

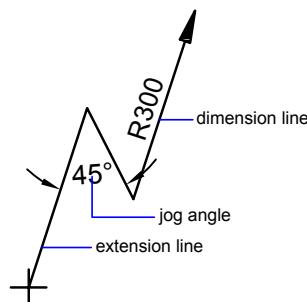
The size of the center mark is the distance from the center of the circle or arc to the end of the center mark.

Create Jogged Radius Dimensions

With **DIMJOGGED**, you can create jogged radius dimensions, also called “foreshortened radius dimensions,” when the center of an arc or circle is located off the layout and cannot be displayed in its true location. The origin point of the dimension can be specified at a more convenient location called the *center location override*.



You can control the default angle of the jog in the Modify Dimension Style dialog box, Symbols and Arrows tab, under Radius Dimension Jog.



Once a jogged radius dimension is created, you can modify the jog and the center location override by

- Using grips to move the features
- Changing the locations of the features with the Properties palette
- Using STRETCH

NOTE Jogged radius dimensions can be viewed but not edited in versions previous to AutoCAD 2006. Also, if you make dramatic changes to the associated geometry, you may get unpredictable results for the jogged radius dimension.

See also:

["Fit Dimension Text Within Extension Lines" on page 889](#)

To create a diameter dimension

- 1 Click Dimension menu ► Diameter.
- 2 Select the arc or circle to dimension.
- 3 Enter options as needed:
 - To edit the dimension text content, enter **t** (Text) or **m** (multiline text). Editing within or overwriting the brackets (<>) changes or removes the dimension value. Adding text before or after the brackets appends text before or after the dimension value.
 - To change the dimension text angle, enter **a** (Angle).
- 4 Specify the leader line location.

Dimension toolbar



Command line: DIMDIAMETER

To create a radius dimension

- 1 Click Dimension menu ► Radius.
- 2 Select an arc, circle, or polyline arc segment.
- 3 Enter options as needed:
 - To edit the dimension text content, enter **t** (Text) or **m** (multiline text). Editing within or overwriting the brackets (<>) changes or removes the dimension value. Adding text before or after the brackets appends text before or after the dimension value.
 - To edit the dimension text angle, enter **a** (Angle).
- 4 Specify the leader line location.

Dimension toolbar



Command line: DIMRADIUS

To create a jogged radius dimension

- 1 Click Dimension menu ► Jogged.
- 2 Select an arc, circle, or polyline arc segment.
- 3 Specify a point for the dimension origin (the center location override).
- 4 Specify a point for the dimension line angle and the dimension text location.
- 5 Specify another point for the location of the dimension jog.

Dimension toolbar



Command line: DIMJOGGED

To create centerlines automatically with radial dimensions

- 1 Click Dimension menu ► Style.
- 2 In the Dimension Style Manager, select the style you want to change. Click Modify.
- 3 In the Modify Dimension Style dialog box, Lines tab, under Center Marks, select Line in the Type box. The example area reflects your selection.
- 4 In the Size box, enter the centerline size.

Dimension or Styles toolbar



Command line: DIMSTYLE

To create centerlines or center marks on an arc or circle

- 1 Click Dimension menu ► Style.
- 2 In the Dimension Style Manager, select the style you want to change. Click Modify.
- 3 In the Modify Dimension Style dialog box, Lines tab, under Center Marks, select Line in the Type box. The example area reflects your selection.
- 4 In the Size box, enter the centerline size.

5 Click Dimension menu ► Center Mark.

6 Select an arc or a circle.

Dimension toolbar



Command line: DIMCENTER

Create Angular Dimensions

Angular dimensions measure the angle between two lines or three points. To measure the angle between two radii of a circle, you select the circle and specify the angle endpoints. With other objects, you select the objects and then specify the dimension location. You can also dimension an angle by specifying the angle vertex and endpoints. As you create the dimension, you can modify the text content and alignment before specifying the dimension line location.

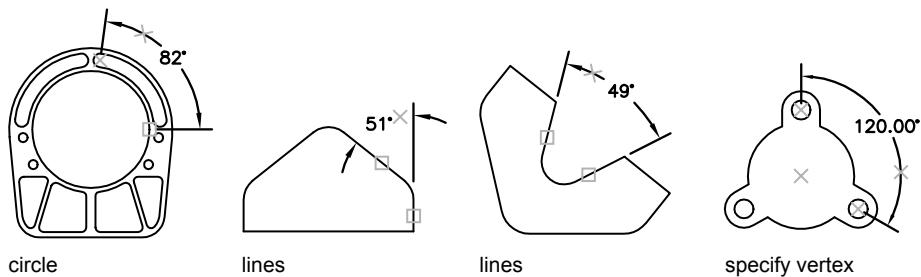
NOTE You can create baseline and continued angular dimensions relative to existing angular dimensions. Baseline and continued angular dimensions are limited to 180 degrees or less. To obtain baseline and continued angular dimensions larger than 180 degrees, use grip editing to stretch the location of the extension line of an existing baseline or continued dimension.

Dimension Lines

If you use two straight, nonparallel lines to specify an angle, the dimension line arc spans the angle between the two lines. If the dimension line arc does not meet one or both of the lines being dimensioned, the program draws one or two extension lines to intersect the dimension line arc. The arc is always less than 180 degrees.

Dimension Circles and Arcs

If you use an arc or a circle or three points to specify an angle, the program draws the dimension line arc between the extension lines. The extension lines are drawn from the angle endpoints to the intersection of the dimension line arc.



The location that you specify for the dimension line arc determines the quadrant of the dimensioned angle.

To create an angular dimension

- 1 Click Dimension menu ▶ Angular.
- 2 Use one of the following methods:
 - To dimension a circle, select the circle at the first endpoint of the angle and then specify the second endpoint of the angle.
 - To dimension any other object, select the first line, and then select the second line.
- 3 Enter options as needed:
 - To edit the dimension text content, enter **t** (Text) or **m** (multiline text). Editing within or overwriting the brackets (<>) changes or removes the calculated dimension value. Adding text before or after the brackets appends text before or after the dimension value.
 - To edit the dimension text angle, enter **a** (Angle).
- 4 Specify the dimension line arc location.

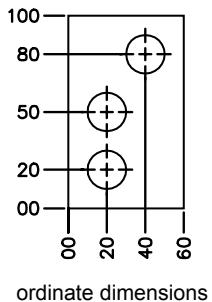
Dimension toolbar



Command line: DIMANGULAR

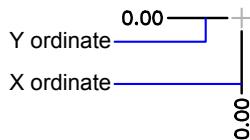
Create Ordinate Dimensions

Ordinate dimensions measure the perpendicular distance from an origin point called the *datum* to a dimensioned feature, such as a hole in a part. These dimensions prevent escalating errors by maintaining accurate offsets of the features from the datum.

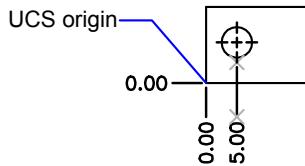


ordinate dimensions

Ordinate dimensions consist of an *X* or *Y* value with a leader line. *X*-datum ordinate dimensions measure the distance of a feature from the datum along the *X* axis. *Y*-datum ordinate dimensions measure the distance along the *Y* axis. If you specify a point, the program automatically determines whether it is an *X*- or *Y*-datum ordinate dimension. This is called an automatic ordinate dimension. If the distance is greater for the *Y* value, the dimension measures the *X* value. Otherwise, it measures the *Y* value.



The program uses the absolute coordinate value of the current UCS to determine the ordinate values. Before creating ordinate dimensions, you typically reset the UCS origin to coincide with the datum.



The dimension text is aligned with the ordinate leader line, regardless of the text orientation defined by the current dimension style. You can accept the default text or supply your own.

To create ordinate dimensions

- 1 Click Dimension menu ► Ordinate.
- 2 If straight ordinate leaders are required, turn Ortho mode on.
- 3 At the Select Feature Location prompt, specify a point location.
- 4 Enter **x** (X Datum) or **y** (Y Datum).
You can skip this step by making sure that the ordinate leader endpoint is close to vertical for an X datum or close to horizontal for a Y datum.
- 5 Specify the ordinate leader endpoint.

Dimension toolbar

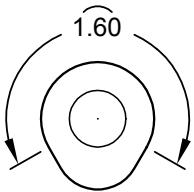


Command line: DIMORDINATE

Create Arc Length Dimensions

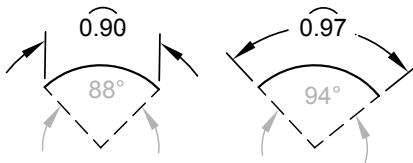
Arc length dimensions measure the distance along an arc or polyline arc segment.

Typical uses of arc length dimensions include measuring the travel distance around a cam or indicating the length of a cable. To differentiate them from linear or angular dimensions, arc length dimensions display an arc symbol by default.



The arc symbol, also called a hat or cap, is displayed either above the dimension text or preceding the dimension text. You can specify the placement style using the Dimension Style Manager. The placement style can be changed on the Symbols and Arrows tab of either the New Dimension Style dialog box or the Modify Dimension Style dialog box.

The extension lines of an arc length dimension can be orthogonal or radial.



NOTE Orthogonal extension lines are displayed only when the included angle of the arc is less than 90 degrees.

To create an arc length dimension

- 1 Click Dimension menu ► Arc Length.
- 2 Select an arc or polyline arc segment.
- 3 Specify the dimension line location.

Dimension toolbar



Command line: DIMARC

Modify Existing Dimensions

You can modify all components of the existing dimension objects in a drawing either individually or by using dimension styles.

Apply a New Dimension Style to Existing Dimensions

You can modify existing dimensions by applying a different dimension style. If you make changes to a dimension style, you can choose whether to update the dimensions associated with that dimension style.

When you create a dimension, the current dimension style is associated with that dimension. The dimension retains this dimension style unless you apply a new dimension style to it or set up dimension style overrides.

You can modify existing dimensions by applying a different dimension style. If you make changes to a dimension style, you can choose whether to update the dimensions associated with that dimension style.

You can restore an existing dimension style or apply the current dimension style, including any dimension style overrides, to selected dimensions.

To apply the current dimension style to existing dimensions

- 1 Click Dimension menu ► Update.
- 2 Select the dimensions to update to the current dimension style.
- 3 Press ENTER.

Dimension toolbar



Command line: DIMSTYLE

To restore a dimension style

- 1 Click Dimension menu ► Style.
- 2 In the Dimension Style Manager, select the dimension style to restore. Click Set Current.
- 3 Click Close.

Dimension toolbar



Command line: DIMSTYLE

Alternate

- On the Styles toolbar, click the arrow in the Dimension Styles control and select a dimension style.

Override a Dimension Style

With dimension style overrides, you can temporarily change a dimensioning system variable without changing the current dimension style.

A dimension style override is a change made to specific settings in the current dimension style. It is equivalent to changing a dimensioning system variable without changing the current dimension style.

You can define dimension style overrides for individual dimensions, or for the current dimension style.

- For individual dimensions, you may want to create overrides to suppress a dimension's extension lines or modify text and arrowhead placement so that they do not overlap drawing geometry without creating a different dimension style.
- You can also set up overrides to the current dimension style. All dimensions you create in the style include the overrides until you delete the overrides, save the overrides to a new style, or set another style current. For example, if you choose Override in the Dimension Style Manager, and change the color of extension lines on the Lines tab, the current dimension style remains unchanged. However, the new value for color is stored in the DIMCLRE system variable. The next dimension you create will have extension lines in the new color. You can save the dimension style overrides as a new dimension style.

Some dimension characteristics are common to a drawing or to a style of dimensioning and are therefore suited to be permanent dimension style settings. Others generally apply on an individual basis and can be applied more effectively as overrides. For example, a drawing usually uses a single type of arrowhead, so it makes sense to define the arrowhead type as part of the dimension style. Suppression of extension lines, however, usually applies in individual cases only and is more suited to a dimension style override.

There are several ways to set up dimension style overrides. You can change options in the dialog boxes or change system variable settings on the command line. You reverse the override by returning the changed settings to their original values. The overrides apply to the dimension you are creating and all subsequent dimensions created with that dimension style until you reverse the override or make another dimension style current.

Example: Change a Dimension Style Override on the Command Line

You can override the current dimension style while creating a dimension by entering the name of any dimensioning system variable at any prompt. In this example, the dimension line color is changed. The change affects subsequent dimensions you create until you reverse the override or make another dimension style current.

Command: **dimlinear**

Specify first extension line origin or <select object>: **dimclrd**

Enter new value for dimension variable <byblock>: **5**

Specify first extension line origin <select object>: *Specify the first extension line origin, or select an object to dimension*

To set up dimension style overrides

- 1 Click Dimension menu ► Style.
- 2 In the Dimension Style Manager, under Styles, select the dimension style for which you want to create an override. Click Override.
- 3 In the Override Current Style dialog box, make changes to the dimension style by clicking the appropriate tab.
- 4 Click OK to return to the Dimension Style Manager.
The dimension style overrides are listed below the style they modify in the dimension style name list.
- 5 Click Close.

Dimension or Styles toolbar



Command line: DIMSTYLE

To apply dimension style overrides

- 1 Click Dimension menu ► Style.
- 2 In the Dimension Style Manager, click Override.

- 3 In the Override Current Style dialog box, enter the style overrides. Click OK.

The program displays <style overrides> below the dimension style name in the Dimension Style Manager dialog box. After you create dimension style overrides, you can continue to modify dimension styles, compare them with other dimension styles, or delete or rename the overrides.

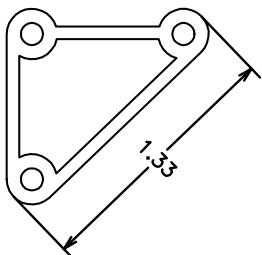
Dimension or Styles toolbar



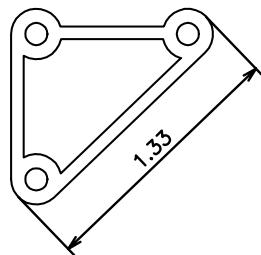
Command line: DIMSTYLE

Modify Dimension Text

Once you've created a dimension, you can rotate the existing text or replace it with new text. You can move the text to a new location or back to its home position, which is the position defined by the current dimension style. In the following illustration, the home position is above and centered on the dimension line.



dimension text rotated

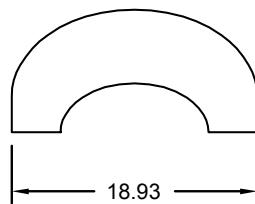


dimension text moved
back to home position

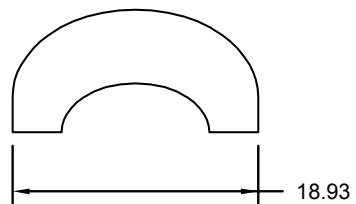
When you rotate or replace dimension text, you specify the change first, for example, rotating the text to be at an angle. When you move dimension text, you select a single dimension to move.

You can move dimension text to the left, right, or center along the dimension line or to any position inside or outside the extension lines. A quick and simple way to do this is by using grips. If you move text up or down, the current vertical alignment of the text relative to the dimension line is not changed, so the dimension and extension lines are modified accordingly. The following

illustration shows the result of moving text down and to the right. The text remains centered vertically in relation to the dimension line.



text centered vertically on
the dimension line



result of moving text to the right
and outside the extension lines

See also:

[“Control Dimension Text” on page 889](#)

To rotate dimension text

- 1 Click Dimension menu > Align Text > Angle.
- 2 Select the dimension to edit.
- 3 Enter the new angle for the text.

Dimension toolbar



Command line: DIMTEDIT

To return dimension text to its home position

- 1 Click Dimension menu > Align Text > Home.
- 2 Select the dimension text you want to return to its home position.

Dimension toolbar



Command line: DIMTEDIT

To replace existing dimension text with new text

- 1 Click Modify menu ► Object ► Text ► Edit.
- 2 Select the dimension text you want to edit.
- 3 In the In-Place Text Editor, enter the new dimension text. Click OK.

Text toolbar



Command line: DDEDIT

To move text to the left side of the dimension line

- 1 Click Dimension menu ► Align Text ► Left.
- 2 Select the dimension.

The dimension text is left-justified along the dimension line inside the extension lines. You can choose the Center or Right options to move the text to the center or right of the dimension line.

Dimension toolbar



Command line: DIMTEDIT

To set dimension line spacing for baseline and continued dimensions

- 1 Click Dimension menu ► Style.
- 2 In the Dimension Style Manager, select the style you want to change. Click Modify.
- 3 In the Modify Dimension Style dialog box, Lines tab, under Dimension Lines, enter in the Baseline Spacing box the offset distance between dimension lines for baseline and continued dimensions.
- 4 Click OK.
- 5 Click Close to exit the Dimension Style Manager.

Dimension or Styles toolbar



Command line: DIMSTYLE

To change the extension origin offset

- 1 Click Dimension menu ► Style.
- 2 In the Dimension Style Manager, select the style you want to change. Click Modify.
- 3 In the Modify Dimension Style dialog box, Lines tab, under Extension Lines, enter the new value for Offset from Origin.
- 4 Click OK.
- 5 Click Close to exit the Dimension Style Manager.

Dimension or Styles toolbar



Command line: DIMSTYLE

Modify Dimension Geometry

Grip editing is the quickest and easiest way to modify the location of dimension elements. How you edit dimensions depends whether the dimension is associative.

You can modify dimensions with the editing commands and with grip editing. Grip editing is the quickest and easiest way to modify dimensions. How you edit dimensions depends on whether the dimension is associative.

Modify Associative Dimensions

Associative dimensions retain their associativity to dimensioned objects through many editing commands if both the dimension and the associated geometry are selected and operated on with a single command. For example, if a dimension and its associated geometry are moved, copied, or arrayed in the same command, each dimension retains associativity with its respective geometry.

In some circumstances, dimensions are automatically disassociated, including

- If the associated geometric object is erased

- If the associated geometric object undergoes a Boolean operation such as UNION or SUBTRACT
- If grip editing is used to stretch a dimension parallel to its dimension line
- If the association to a geometric object is specified using the Apparent Intersection object snap, and the geometric object is moved so that the apparent intersection no longer exists

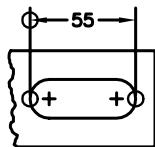
In other circumstances, a dimension may become partially associated. For example, if a linear dimension is associated with the endpoints of two geometric objects and one of the objects is erased, the remaining association is preserved. The disassociated end of the linear dimension may then be associated with another geometric object using DIMREASSOCIATE.

NOTE The command line displays a warning message if a dimension is disassociated.

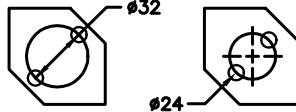
Modify Nonassociative Dimensions

For nonassociative dimensions, when you edit dimensioned objects, you must include the relevant dimension definition points in the selection set, or the dimension is not updated. Definition points determine the dimension location. For example, to stretch a dimension, you must include the appropriate definition points in the selection set. You can easily include them by turning on grips and selecting the object so that the grips are highlighted.

The definition points for each type of dimension are indicated in the following illustrations. The middle point of the dimension text is a definition point for all dimension types.



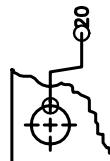
Linear: extension line origins and intersection of first extension line and dimension line



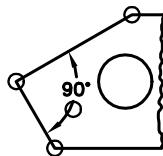
Diameter: selection point and opposite point



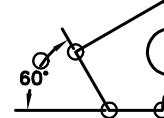
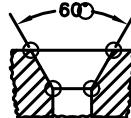
Radius: selection point and center



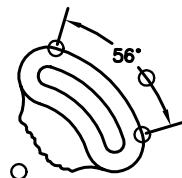
ordinate: feature location and leader endpoint



Three-Point Angular: angle vertex, extension line origins, and dimension line arc



Two-Point Angular: extension line origins, and dimension line arc



If no angle vertex is shown, definition points are placed at the ends of the lines that form the angle. In the two-line angular example, a definition point is placed at the center point of the dimensioned arc.

NOTE Definition points are drawn on a special layer named DEFPOINTS, which is not plotted.

Modify Exploded Dimensions

You can edit exploded dimensions as you would any other objects because an exploded dimension is a collection of separate objects: lines, 2D solids, and text. Occasionally you may need to explode a dimension to make changes

such as creating a break in a dimension line or extension line. Once a dimension is exploded, you cannot reassociate the dimension into a dimension object.

See also:

[“Control Dimension Geometry” on page 883](#)

Change Dimension Associativity

You may need to change the associativity of dimensions in several circumstances including adding associativity to dimensions created in previous releases.

You may need to change the associativity of dimensions in several circumstances such as the following:

- Redefine the associativity of dimensions in drawings that have been edited significantly.
- Add associativity to dimensions that have been partially disassociated.
- Add associativity to dimensions in legacy drawings.
- Remove associativity from dimensions in drawings that will be used by people working in releases prior to AutoCAD 2002, but who do not want any proxy objects in the drawings.

Reassociate Dimensions to Different Objects

With DIMREASSOCIATE, you can select one or more dimensions and step through the extension-line origin points of each dimension. For each extension-line origin point, you can specify a new *association point* on a geometric object. Association points determine the attachment of extension lines to locations on geometric objects.

NOTE When you create or modify associative dimensions, it is important to locate their association points carefully so that if you make a future design change, the geometric objects that you change will also change the dimensions associated with them.

When you use the DIMREASSOCIATE command, a marker is displayed that indicates whether each successive extension line origin point of the dimension is associative or nonassociative. A square with an X in it means that the point is associated with a location on an object, while an X without the square means that the point is not associated with an object. Use an object snap to specify the new association for the extension-line origin point or press ENTER to skip to the next extension-line origin point.

NOTE The marker disappears if you pan or zoom with a wheel mouse.

Change Nonassociative Dimensions to Associative

You can change all the nonassociative dimensions in a drawing to associative. Use QSELECT to select all nonassociative dimensions, and then use DIMREASSOCIATE to step through the dimensions, associating each one with locations on geometric objects.

Change Associative Dimensions to Nonassociative

You can change all associative dimensions in a drawing to nonassociative dimensions. Use QSELECT to select all associative dimensions, and then use DIMDISASSOCIATE to convert them into nonassociative dimensions.

See also:

["Associative Dimensions" on page 878](#)

["Save Drawings to Previous Drawing File Formats"](#)

To associate or reassociate a dimension

- 1 Click Dimension menu ► Reassociate Dimensions.
- 2 Select one or more dimensions to associate or reassociate.
- 3 Do one of the following:
 - Specify the new location of the extension-line origin point.
 - Enter **s** and select a geometric object to associate with the dimension.
 - Press ENTER to skip to the next extension-line origin point.
 - Press ESC to end the command but keep any associations you made up to that point.
- 4 Repeat the previous step as needed.

Command line: DIMREASSOCIATE

To disassociate a dimension

- 1 On the command line, enter **DIMDISASSOCIATE**.
- 2 Select one or more dimensions to disassociate and press ENTER when you finish.

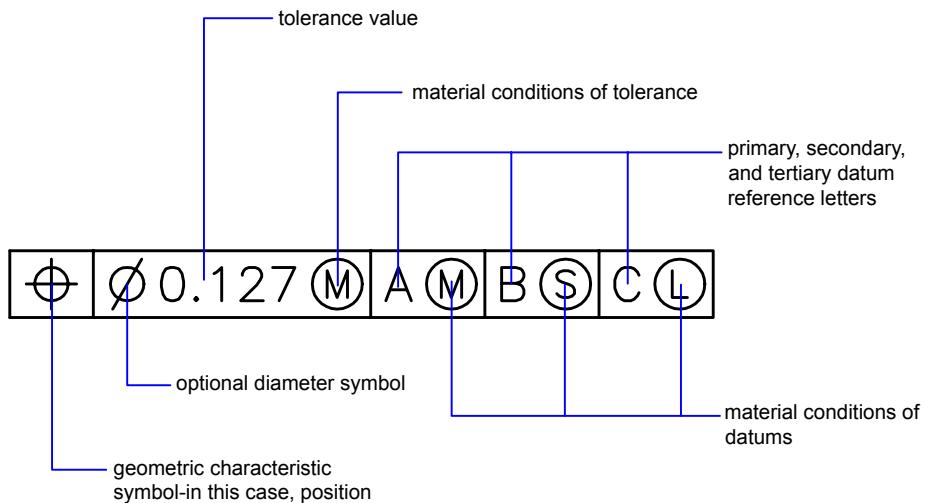
Add Geometric Tolerances

You can add geometric tolerances that show acceptable deviations of form, profile, orientation, location, and runout of a feature.

Overview of Geometric Tolerances

Geometric tolerances show acceptable deviations of form, profile, orientation, location, and runout of a feature. You add geometric tolerances in feature control frames. These frames contain all the tolerance information for a single dimension. Geometric tolerances can be created with or without leader lines, depending on whether you create them with 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.

To create geometric tolerances

- 1 Click Dimension menu ► Tolerance.
- 2 In the Geometric Tolerance dialog box, click the first square under Sym and select a symbol to insert.
- 3 Under Tolerance 1, click the first black box to insert a diameter symbol.
- 4 In the Text box, enter the first tolerance value.
- 5 To add a material condition (optional), click the second black box and click a symbol in the Material Conditions dialog box to insert it.
- 6 In the Geometric Tolerance dialog box, add a second tolerance value (optional) in the same way as the first tolerance value.
- 7 Under Datum 1, Datum 2, Datum 3, enter the datum reference letter.
- 8 Click the black box to insert a material condition symbol for each datum reference.
- 9 In the Height box, enter a height.
- 10 Click the Projected Tolerance Zone box to insert the symbol.
- 11 In the Datum Identifier box, add a datum value.
- 12 Click OK.
- 13 In the drawing, specify a location for the feature control frame.

Dimension toolbar



Command line: TOLERANCE

To create a geometric tolerance with a leader

- 1 On the command line, enter **leader**.
- 2 Specify the start point of the leader.
- 3 Specify the second point of the leader.
- 4 Press ENTER twice to display the Annotation options.
- 5 Enter **t** (Tolerance), and create a feature control frame.

The feature control frame is attached to the endpoint of the leader.

Command line: LEADER

Material Conditions

Material conditions apply to features that can vary in size.

The second compartment contains the tolerance value. Depending on the control type, the tolerance value is preceded by a diameter symbol and followed by a material condition symbol.

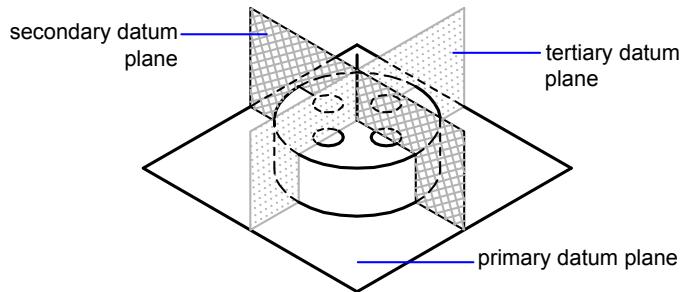
Material conditions apply to features that can vary in size:

- At *maximum material condition* (symbol M, also known as MMC), a feature contains the maximum amount of material stated in the limits.
- At MMC, a hole has minimum diameter, whereas a shaft has maximum diameter.
- At *least material condition* (symbol L, also known as LMC), a feature contains the minimum amount of material stated in the limits.
- At LMC, a hole has maximum diameter, whereas a shaft has minimum diameter.
- *Regardless of feature size* (symbol S, also known as RFS) means a feature can be any size within the stated limits.

Datum Reference Frames

The tolerance values in the feature control frame are followed by up to three optional datum reference letters and their modifying symbols. A datum is a theoretically exact point, axis, or plane from which you make measurements and verify dimensions. Usually, two or three mutually perpendicular planes perform this task best. These are jointly called the datum reference frame.

The following illustration shows a datum reference frame verifying the dimensions of the part.



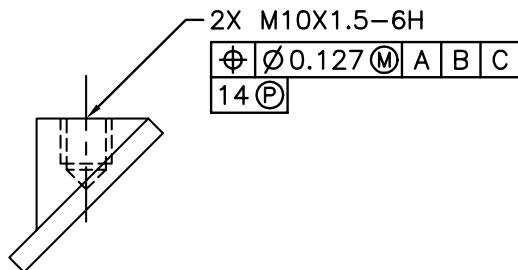
Projected Tolerance Zones

Projected tolerances are specified in addition to positional tolerances to make the tolerance more specific. For example, projected tolerances control the perpendicularity tolerance zone of an embedded part.

The symbol for projected tolerance (

\circ

) is preceded by a height value, which specifies the minimum projected tolerance zone. The projected tolerance zone height and symbol appear in a frame below the feature control frame, as shown in the following illustration.

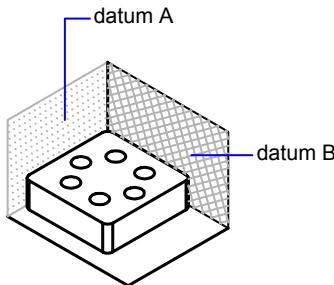


Composite Tolerances

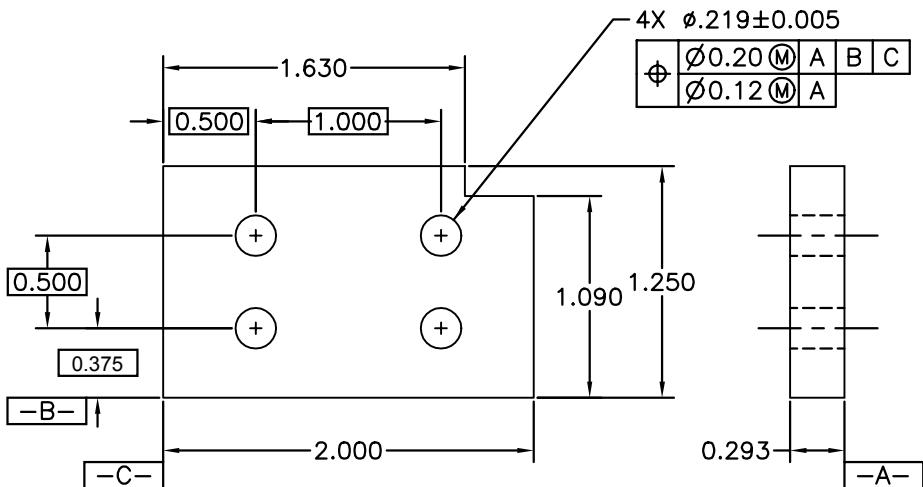
A composite tolerance specifies two tolerances for the same geometric characteristic of a feature or for features that have different datum requirements. One tolerance relates to a pattern of features and the other

tolerance to each feature within the pattern. The individual feature tolerance is more restrictive than the pattern tolerance.

In the following illustration, the point where datums A and B intersect is called the datum axis, the point from which the position of the pattern is calculated.



A composite tolerance could specify both the diameter of the pattern of holes and the diameter of each individual hole, as in the following illustration.



When you add composite tolerances to a drawing, you specify the first line of a feature control frame and then choose the same geometric characteristic symbol for the second line of the feature control frame. The geometric symbol

compartment is extended over both lines. You can then create a second line of tolerance symbols.

Part 8

Plot and Publish Drawings

Chapter 29 Prepare Drawings for Plotting and Publishing

Chapter 30 Plot Drawings

Chapter 31 Publish Drawings

29

Prepare Drawings for Plotting and Publishing

You prepare your drawing for plotting or publishing by specifying page setup settings. These settings are stored in the drawing file with the layout. Once a layout is established, you can modify the settings of its page setup or apply a different page setup.

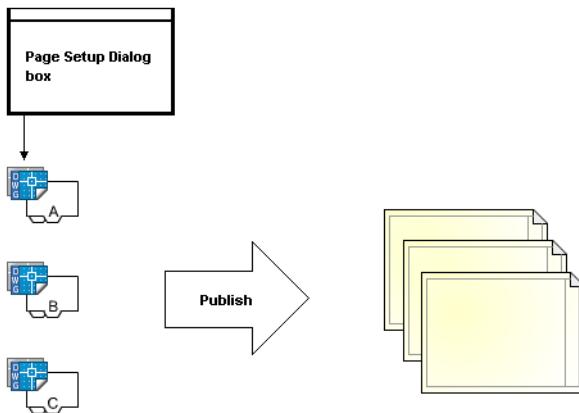
In this chapter

- Quick Start to Preparing Drawings for Plotting and Publishing
- Specify Page Setup Settings
- Use the Layout Wizard to Specify Layout Settings
- Import PCP or PC2 Settings into a Layout
- Create and Use Named Page Setups
- Use Named Page Setups with Sheet Sets

Quick Start to Preparing Drawings for Plotting and Publishing

Preparing a drawing for plotting or publishing requires specifying many settings and options that define the output of your drawing. These settings can be saved as a *named page setup*.

You can apply a named page setup to multiple layouts using the Page Setup Manager. You can also import a named page setup from another drawing and apply it to layouts in the current drawing.



If you want to specify the same output options for several layouts or sheets, use named page setups.

Specify Page Setup Settings

Page setups are associated with layouts and stored in the drawing file. The settings specified in a page setup determine the appearance and format of your final output.

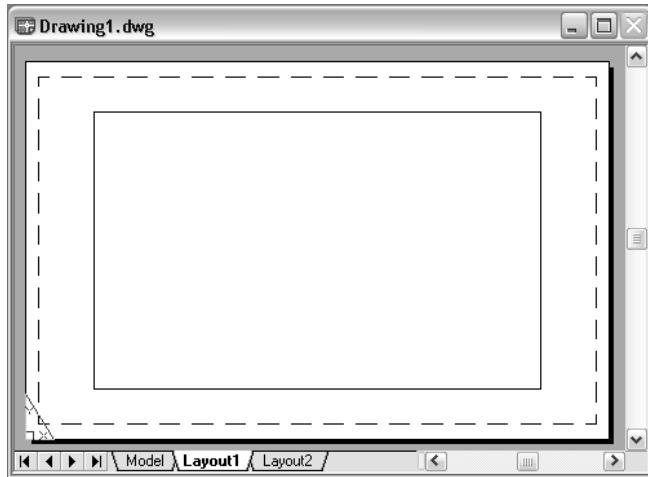
Overview of Page Setup Settings

A page setup is a collection of plot device and other settings that affect the appearance and format of your final output. These settings can be modified and applied to other layouts.

After you complete a drawing on the Model tab, you can begin creating a layout to plot by clicking a layout tab. When you click a layout tab for the

first time, a single viewport is displayed on the page. A dashed line indicates the printable area of the paper for the currently configured paper size and plotter.

Once you have your layout set up, you specify the settings for the layout's page setup, which includes the plot device settings and other settings that affect the appearance and format of the output. The settings you specify in the page setup are stored in the drawing file with the layout. You can modify the settings of a page setup at any time.



By default, every initialized layout has a page setup associated with it. A layout is initialized if its paper size is defined in the page setup to any size other than 0 x 0. You can apply a named page setup saved with one layout to another layout. This creates a new page setup with the same settings as the first one.

If you want the Page Setup Manager to be displayed each time you begin a new drawing layout, select the Show Page Setup Manager for New Layouts option on the Display tab in the Options dialog box. If you don't want a viewport to be automatically created for each new layout, clear the Create Viewport in New Layouts option on the Display tab in the Options dialog box.

To modify the settings of a layout's page setup

- 1 Click the layout tab for which you want to modify the page setup settings.
- 2 Click File menu ► Page Setup Manager.
- 3 In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.

- 4** Click Modify.
- 5** In the Page Setup dialog box, modify the required settings. Click OK.
- 6** In the Page Setup Manager, click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

To apply a layout's named page setup to another layout

- 1** Click the layout tab to which you want to apply another layout's page setup settings.
- 2** Click File menu ► Page Setup Manager.
- 3** In the Page Setup Manager, Page Setups area, select a named page setup that you want to apply to the layout selected in the drawing area.
- 4** Click Set Current.
- 5** Click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

Select a Printer or Plotter for a Layout

When you create a layout, you must select a printing or plotting device in the Page Setup dialog box in order for the layout to be printed or plotted. Once you've selected a device, you can view details about the name and location of the device, and you can change the device's configuration.

The printer or plotter you select in the Page Setup dialog box determines the printable area of the layout. This printable area is indicated by the dashed line in the layout. If you change the paper size or the printing or plotting device, it may change the printable area of your drawing page.

See also:

[“Select a Printer or Plotter” on page 980](#)

[“Control PC3 File Device and Document Settings” in the *Driver and Peripheral Guide*](#)

To select a printer or plotter for a layout

- 1 Click the layout tab for which you want to specify a printer or plotter.
- 2 Click File menu ► Page Setup Manager.
- 3 In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
- 4 Click Modify.
- 5 In the Page Setup dialog box, under Printer/Plotter, select a printer or plotter from the list. Click OK.
- 6 In the Page Setup Manager, click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab and select Page Setup Manager.

To change the configuration of a printer or plotter specified in a page setup

- 1 Click the layout tab for which you want to specify a printer or plotter.
- 2 Click File menu ► Page Setup Manager.
- 3 In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
- 4 Click Modify.
- 5 In the Page Setup dialog box, under Printer/Plotter, click Properties.
- 6 In the Plotter Configuration Editor, change the required settings. Click OK.
- 7 In the Page Setup dialog box, click OK.
- 8 In the Page Setup Manager, click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

Select a Paper Size for a Layout

You can select a paper size from a standard list. The paper sizes available in the list are determined by the plot device that is currently selected for the layout. If your plotter is configured for raster output, you must specify the output size in pixels. You can add custom paper sizes, which are stored in the plotter configuration (PC3) file, using the Plotter Configuration Editor.

If you are using a system printer, the paper size is determined by the document defaults that are set in the Windows Control Panel. The default paper size is displayed in the Page Setup dialog box when you create a new layout for that configured device. If you change the paper size in the Page Setup dialog box, the new paper size is saved with the layout and overrides the size saved in the plotter configuration (PC3) file.

See also:

[“Control PC3 File Device and Document Settings” in the *Driver and Peripheral Guide*](#)

To set the paper size for a layout

- 1 Click the layout tab for which you want to set the paper size.
- 2 Click File menu ▶ Page Setup Manager.
- 3 In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
- 4 Click Modify.
- 5 In the Page Setup dialog box, under Paper Size, select a paper size from the list. Click OK.
- 6 In the Page Setup Manager, click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

To start the Plotter Configuration Editor

- 1 Click File menu ► Plotter Manager.
- 2 In the Plotter Manager, double-click the plotter configuration (PC3) file you want to edit.

The Plotter Configuration Editor is displayed.

Command line: PLOTTERMANAGER

To add a custom paper size from scratch

- 1 Click File menu ► Plotter Manager.
- 2 In the Plotter Manager, double-click the plotter configuration (PC3) file you want to edit.
- 3 In the Plotter Configuration Editor, Device and Document Settings tab, double-click User-Defined Paper Sizes & Calibration to display the calibration and paper size options.
- 4 Select Custom Paper Sizes.
- 5 Under Custom Paper Sizes, click Add.
- 6 In the Custom Paper Size wizard, Begin page, select Start from Scratch. Click Next.
- 7 On the Media Bounds page, in the Units list, select either Inches or Millimeters for paper size.
When a nondimensional raster image, such as BMP or TIFF, is plotted, the size of the plot is specified in pixels, not inches or millimeters.
- 8 In the Width and Length lists, specify the paper width and length. Click Next.

NOTE Each plotter has a maximum printable area determined by where it grips the paper and how far the pen shuttle reaches. If you are creating a paper size that is larger than the paper sizes offered in the Custom Paper Size wizard, verify that the plotter is capable of plotting the new dimensions.

- 9 On the Printable Area page, use Top, Bottom, Left, and Right to specify the printable area. Click Next.
- 10 On the Paper Size Name page, enter a name for the paper size. Click Next.

- 11 On the File Name page, enter a name for the PMP file.
- 12 On the Finish page, specify whether the paper source is Sheet-Fed or Roll-Fed.
- 13 Click Print Test Page to verify the custom size.
A cross is printed that defines the paper size and a rectangle that defines the printable area. If all four sides of the rectangle are not printed, increase the printable area.
- 14 Click Finish to exit the Custom Paper Size wizard.

Command line: PLOTTERMANAGER

To add a new custom paper size starting from an existing paper size

- 1 Click File menu ► Plotter Manager.
- 2 In the Plotter Manager, double-click the plotter configuration (PC3) file you want to edit.
- 3 In the Plotter Configuration Editor, Device and Document Settings tab, double-click User-Defined Paper Sizes & Calibration to display the calibration and paper size settings.
- 4 Select Custom Paper Sizes.
- 5 Under Custom Paper Sizes, click Add.
- 6 In the Custom Paper Size wizard, Begin page, select Use Existing.
- 7 In the list of existing standard paper sizes, select a paper size on which to base the custom paper size you are creating.
- 8 Follow the instructions in “To add a custom paper size from scratch” on page 951 to continue through the Custom Paper Size wizard.
The new paper size is a user-defined size, not a standard size.
- 9 Click Finish to exit the Custom Paper Size wizard.

Command line: PLOTTERMANAGER

To edit a custom paper size

- 1 Click File menu ► Plotter Manager.
- 2 In the Plotter Manager, double-click the plotter configuration (PC3) file you want to edit.

- 3** In the Plotter Configuration Editor, Device and Document Settings tab, double-click User-Defined Paper Sizes & Calibration to display the calibration and paper size settings.
- 4** Select Custom Paper Sizes.
- 5** Under Custom Paper Sizes, select a paper size from the list. Click Edit.
- 6** In the Custom Paper Size wizard, make changes to the paper size, printable area, custom paper size name, and source.
- 7** Click Finish to exit the Custom Paper Size wizard.

Command line: PLOTTERMANAGER

To delete a custom paper size

- 1** Click File menu ► Plotter Manager.
- 2** In the Plotter Manager, double-click the plotter configuration (PC3) file you want to edit.
- 3** In the Plotter Configuration Editor, Device and Document Settings tab, double-click User-Defined Paper Sizes & Calibration to display the calibration and paper size settings.
- 4** Click Custom Paper Sizes.
- 5** Under Custom Paper Sizes, select a paper size from the list.
- 6** Click Delete.

Command line: PLOTTERMANAGER

To modify a standard paper size

- 1** Click File menu ► Plotter Manager.
- 2** In the Plotter Manager, double-click the Plotter Configuration (PC3) file you want to edit.
- 3** In the Plotter Configuration Editor, Device and Document Settings tab, double-click User-Defined Paper Sizes & Calibration to display the calibration and paper size settings.
- 4** Select Modify Standard Paper Sizes.
- 5** Under Modify Standard Paper Sizes, select the paper size you want to adjust. Click Modify.

- 6 In the Custom Paper Size wizard, adjust the printable area as necessary. Click Finish to exit the Custom Paper Size wizard.

Command line: PLOTTERMANAGER

Set the Plot Area of a Layout

When you prepare to plot from the Model tab or a layout tab, you can specify the plot area to determine what will be included in the plot. When you create a new layout, the default Plot Area option is Layout. Layout plots all objects within the printable area of the specified paper size.

The Display Plot Area option plots all the objects displayed in the drawing. The Extents Plot Area option plots all the visible objects in the drawing. The View Plot Area option plots a saved view. You can use the Window Plot Area option to define an area to be plotted.

See also:

["Specify the Area to Plot" on page 980](#)

To set the plot area and adjust the display

- 1 Click the layout tab for which you want to set the plot area and adjust the display.
- 2 Click File menu ► Page Setup Manager.
- 3 In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
- 4 Click Modify.
- 5 In the Page Setup dialog box, under Plot Area, select one of the following options:
 - **Layout.** Prints all objects within the printable area of the paper. This option is only available from a layout tab.
 - **Limits.** Prints or plots the current grid limits. This option is only available from the Model tab.
 - **Extents.** Plots all objects in the drawing.
 - **Display.** Plots all objects displayed in the drawing area.
 - **View.** Prints or plots a saved view. Select a named view from the list provided.

■ **Window.** Plots objects in the area you define. Select the Window option, and then respond to the prompts to define the area. Click the Window button to edit the defined area.

6 Click OK.

7 In the Page Setup Manager, click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

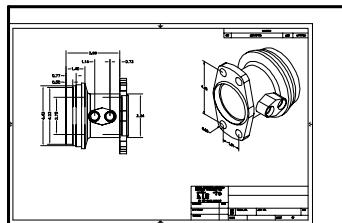
Adjust the Plot Offset of a Layout

The printable area of a drawing sheet is defined by the selected output device and is represented by the dashed line in a layout. When you change the output device, the printable area may change.

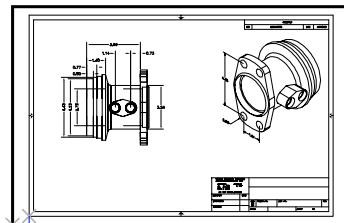
The plot offset specifies an offset of the plot area relative to the lower-left corner (the origin) of the printable area or the edge of the paper, depending on the Specify Plot Offset Relative To option specified in the Options dialog box, Plot and Publish tab. The Plot Offset area of the Plot dialog box displays the specified plot offset option in parentheses.

You can offset the geometry on the paper by entering a positive or negative value in the X and Y Offset boxes. However, this may result in the plot area being clipped.

If you choose to plot an area other than the entire layout, you can also center the plot on the sheet of paper.



plot with origin 0,0



home position
lower-left corner of paper

plot with origin -1.0,-0.5

To adjust the plot offset of a layout

- 1 Click the layout tab for which you want to adjust the plot offset.
- 2 Click File menu ► Page Setup Manager.
- 3 In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
- 4 Click Modify.
- 5 In the Page Setup dialog box, under Plot Offset, enter a value in units for X or Y or both. Click OK.
- 6 In the Page Setup Manager, click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab. Click Page Setup Manager.

Set the Plot Scale for a Layout

When you plot a drawing layout, you can either specify a precise scale for the layout or fit the image to the paper.

Normally, you plot a layout at a 1:1 scale. To specify a different scale for the layout, set the plot scale for the layout in the Page Setup or the Plot dialog box. In those dialog boxes, you can select a scale from a list or enter a scale.

NOTE You can modify the list of scales with 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 223](#)

["Draw, Scale, and Annotate in Model Space" on page 204](#)

To set the plot scale in a layout

- 1 Click the layout tab for which you want to set the plot scale.
- 2 Click File menu ➤ Page Setup Manager.
- 3 In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
- 4 Click Modify.
- 5 In the Page Setup dialog box, under Plot Scale, select a scale from the Scale list.
The default scale when plotting a layout is 1:1. To set a custom plot scale, enter values in the Inches or Millimeters box and the Units box. The type of unit is determined by the paper size, but you can change it in the list box.
- 6 Click OK.
- 7 In the Page Setup Manager, click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab. Click Page Setup.

To set the Fit to Paper option when plotting

- 1 Click the layout tab for which you want to set the plot scale to Fit to Paper.

- 2** Click File menu ► Page Setup Manager.
- 3** In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
- 4** Click Modify.
- 5** In the Page Setup dialog box, Under Plot Scale, select Fit to Paper.

NOTE If the Plot Area is set to Layout, you cannot select the Fit to Paper option.

- 6** Click OK.
- 7** In the Page Setup Manager, click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab. Click Page Setup Manager

Set the Lineweight Scale for a Layout

You can scale lineweights proportionately in a layout with the plot scale.

Typically, lineweights specify the line width of plotted objects and are plotted with the line width size regardless of the plot scale. Most often, you use the default plot scale of 1:1 when plotting a layout. However, if you want to plot an E-size layout that is scaled to fit on an A-size sheet of paper, for example, you can specify lineweights to be scaled in proportion to the new plot scale.

See also:

[“Control Lineweights” on page 317](#)

To scale lineweights in a layout

- 1** Click the layout tab for which you want to scale lineweights.
- 2** Click File menu ► Page Setup Manager.
- 3** In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
- 4** Click Modify.
- 5** In the Page Setup dialog box, under Plot Scale, select Scale Lineweights.

The linewidths in the current layout are scaled in proportion to the designated plot scale. When you are working in the Model tab, this option is not available.

- 6 Click OK.
- 7 In the Page Setup Manager, click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab. Click Page Setup Manager

Select a Plot Style Table for a Layout

A plot style table is a collection of plot styles assigned to a layout or the Model tab. A plot style is an object property, similar to linetype and color. A plot style can be assigned to an object or assigned to a layer. A plot style controls an object's plotted properties.

You can also create a new plot style table to save in the page setup for the layout or edit an existing plot style table.

If you select the Display Plot Styles option under Plot Style Table (Pen Assignments), the properties of the plot styles assigned to objects are displayed in the selected layout.

See also:

["Control How Objects Are Plotted" on page 985](#)

To select a plot style table for a layout

- 1 Click the layout tab for which you want to select a plot style table.
- 2 Click File menu ➤ Page Setup Manager.
- 3 In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
- 4 Click Modify.
- 5 In the Page Setup dialog box, Under Plot Style Table, select a Plot Style Table from the list.
- 6 Click OK.

7 In the Page Setup Manager, click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab. Click Page Setup Manager

To create a new plot style table for a layout

- 1** Click the layout tab for which you want to create a new plot style table.
- 2** Click File menu ► Page Setup Manager.
- 3** In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
- 4** Click Modify.
- 5** In the Page Setup dialog box, Under Plot Style Table, select New from the list.
- 6** Follow the instructions in the wizard that is shown (the Add Color-Dependent Plot Style Table wizard or the Add Named Plot Style Table wizard).
- 7** In the Page Setup dialog box, click OK.
- 8** In the Page Setup Manager, click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab. Click Page Setup Manager

To edit a plot style table for a layout

- 1** Click the layout tab for which you want to edit a plot style table.
- 2** Click File menu ► Page Setup Manager.
- 3** In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
- 4** Click Modify.

- 5 In the Page Setup dialog box, under Plot Style Table, select the plot style table you want to edit from the list.
- 6 Click the Edit button.
- 7 In the Plot Style Table Editor, make the required changes. Click Save & Close.
- 8 In the Page Setup dialog box, click OK.
- 9 In the Page Setup Manager, click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab. Click Page Setup Manager

To display plot styles in a layout

- 1 Click the layout tab for which you want to display plot styles.
- 2 Click File menu ➤ Page Setup Manager.
- 3 In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
- 4 Click Modify.
- 5 In the Page Setup dialog box, under Plot Style Table, select the Display Plot Styles option.
- 6 Click OK.
- 7 In the Page Setup Manager, click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab. Click Page Setup Manager

Set Shaded Viewport and Plot Options for a Layout

Shaded viewport and plot options affect how objects are plotted. The options for shaded viewport plotting give you a large degree of flexibility in conveying your three-dimensional designs to others. You can convey your design intent by choosing how viewports are plotted and by specifying resolution levels.

Shaded Viewport Plotting Options

With shaded plotting options, you can choose whether to plot a set of shaded objects using the As Displayed, Wireframe, Hidden, or Rendered option.

Shaded viewport plotting options apply to all objects in viewports and model space. If you use the Shaded or Rendered options, plot style tables included in the page setup do not affect plots. If you use the Render option, two-dimensional wireframe objects, such as lines, arcs, and text, are not plotted.

NOTE Shaded viewport plotting requires a raster-capable device. Most modern plotters and printers are raster-capable devices.

Plot Options

The following options that can be specified for layouts affect how objects are plotted.

- **Plot Object Lineweights.** Specifies that lineweights assigned to objects and layers are plotted.
- **Plot with Plot Styles.** Specifies that the drawing is plotted using plot styles. Selecting this option automatically plots lineweights. If you do not select this option, objects are plotted with their assigned properties and not with the plot style overrides.
- **Plot Paper Space Last.** Specifies that objects in model space are plotted before those in paper space.
- **Hide Paperspace Objects.** Specifies whether the Hide operation applies to objects in the paper space viewport. This option is available only from a layout tab. The effect of this setting is reflected in the plot preview, but not in the layout.

See also:

- “Set Shaded Viewport Options” on page 988
- “Set Options for Plotted Objects” on page 990

To set shaded viewport options for a layout

- 1 Click the layout tab for which you want to set shaded viewport options.
- 2 Click File menu ► Page Setup Manager.
- 3 In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
- 4 Click Modify.
- 5 In the Page Setup dialog box, under Shaded Viewport Options, select the required settings.
- 6 Click OK.
- 7 In the Page Setup Manager, click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab. Click Page Setup Manager

To set plot options for a layout

- 1 Click the layout tab for which you want to set plot options.
- 2 Click File menu ► Page Setup Manager.
- 3 In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
- 4 Click Modify.
- 5 In the Page Setup dialog box, under Plot Options, select the required settings.
- 6 Click OK.
- 7 In the Page Setup Manager, click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab. Click Page Setup Manager

Determine the Drawing Orientation of a Layout

You can specify the orientation of the drawing on the paper using the Landscape and Portrait settings. Landscape orients the drawing on the paper so that the long edge of the paper is horizontal, and Portrait orients the paper so that the short edge is horizontal. Changing the orientation creates the effect of rotating the paper underneath the drawing.

In either landscape or portrait orientation, you can select Plot Upside-Down to control whether the top or bottom of the drawing is plotted first.

Although you can specify the drawing orientation in both the Page Setup dialog box and the Plot dialog box, the Page Setup settings are always saved and reflected in the layout. In the Plot dialog box, you can override the page setup settings for a single plot; however, the settings you apply are not saved in the layout. To save the settings you apply using the Plot dialog box, click the Apply to Layout button in the Plot dialog box.

If you change the drawing orientation, the layout origin remains in the lower-left corner of the rotated page.

To set the orientation of the plotted drawing

- 1 Click the layout tab for which you want to set the drawing orientation.
- 2 Click File menu ➤ Page Setup Manager.
- 3 In the Page Setup Manager, Page Setups area, select the page setup that you want to modify.
- 4 Click Modify.
- 5 In the Page Setup dialog box, under Drawing Orientation, do one of the following:
 - If your drawing is horizontal, select Landscape.
 - If your drawing is vertical, select Portrait.
 - To rotate 180 degrees, select either Portrait or Landscape, and then select Plot Upside-Down.
- 6 Click OK.
- 7 In the Page Setup Manager, click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab. Click Page Setup Manager

Use the Layout Wizard to Specify Layout Settings

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

- A name for the new layout
- The printer associated with the layout
- A paper size to use for the layout
- The orientation of the drawing on the paper
- A title block
- Viewport setup information
- A location for the viewport configuration in the layout

You can edit the information entered in the wizard later by selecting the layout, clicking Page Setup Manager on the File menu, and clicking Modify in the Page Setup Manager.

To create a layout using the wizard

- 1 Click Insert menu ▶ Layout ▶ Layout Wizard.
- 2 On each page of the Create Layout wizard, select the appropriate settings for the new layout.

When finished, the new layout will be the current layout tab.

Command line: LAYOUTWIZARD

Import PCP or PC2 Settings into a Layout

If you work with drawings created in AutoCAD Release 14 or earlier, you can choose to import layout and plot settings contained in a PCP or PC2 file and

apply them to the current layout. Settings that are saved in a PCP or PC2 file include

- Plot Area
- Rotation
- Paper Size
- Plot Scale
- Plot Origin
- Plot Offset

In addition, a PC2 file contains any resolution information that has been modified by a plotter calibration. Pen assignment information can also be imported and saved in a plot style table using the Add Plot Style Table wizard.

To import plotting device and pen settings information, you can use the Import PCP or PC2 Plot Settings wizard to choose a PCP or PC2 file whose settings you want to import. You can also choose to modify any of the imported settings using the Page Setup dialog box.

To import PCP or PC2 settings into the current layout

- 1 At the Command prompt, enter **pcinwizard**.
- 2 In the Import PCP or PC2 Plot Settings wizard, select the PCP or PC2 file whose settings you want to import into the current layout.

Create and Use Named Page Setups

You can save plot device and other page setup settings as named page setups that can be modified and imported into other drawings.

You can create named page setups and apply them to other layouts in your drawing. Named page setups are saved in the drawing file and can be imported into other drawing files and applied to other layouts. You can also modify the settings of named page setups. If you modify a named page setup, you can choose whether the modifications apply to the current layout or to all the layouts in the current drawing that use the named page setup.

If you want to plot the same layout more than one way, or if you want to specify the same output options for several layouts, use named page setups.

You can apply different named page setups to the same layout to achieve specific results when plotting. For example, you might create the named page setups in the following table to control scaling and paper size.

Page setup name	Description
NoScaling	Plot at scale 1:1, E-size sheet
Scale 1 to 2	Plot at scale 1:2, C-size sheet
Draft	Plot to the draft-quality plotter
Final	Plot to the high-quality plotter
Fit-to-Paper	Fit to Paper, A-size sheet

Once you specify a named page setup for a layout, whenever you plot or publish the layout, it is plotted or published with the settings specified in the named page setup set for the layout.

To create a new named page setup

- 1 Click File menu ➤ Page Setup Manager.
- 2 In the Page Setup Manager, Page Setups area, click New.
- 3 In the New Page Setup dialog box, enter a name for the new page setup.
- 4 Under Start With, select a page setup in the list. The settings specified in the selected page setup will be displayed in the Page Setup dialog box after you click OK.
- 5 Click OK.
- 6 In the Page Setup dialog box, change any required settings. Click OK.
The new page setup is displayed in the Page Setups list in the Page Setup Manager.
- 7 To apply the new page setup to the current layout, in the Page Setup Manager, click Set Current.
- 8 In the Page Setup Manager, click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab. Click Page Setup Manager

To apply a named a page setup to a layout

- 1 Click the layout tab for which you want to apply a named page setup.

- 2** Click File menu ► Page Setup Manager.
- 3** In the Page Setup Manager, under Page Setups, select a named page setup from the list.

NOTE A page setup that is not named is shown in the Page Setup Manager with an asterisk on either side of its layout name. An unnamed page setup does not have the same functionality as a named page setup. For example, if you set an unnamed page setup as the current page setup for another layout, the values are applied to the layout. If you make changes to the original unnamed page setup, those changes do not carry over to the layout to which the unnamed page setup was applied.

- 4** Click Set Current.
- 5** Click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab. Click Page Setup Manager

To modify a named a page setup

- 1** Click File menu ► Page Setup Manager.
- 2** In the Page Setup Manager, under Page Setups, select a named page setup from the list.

NOTE A page setup that is not named is shown in the Page Setup Manager with an asterisk on either side of its layout name. An unnamed page setup does not have the same functionality as a named page setup. For example, if you set an unnamed page setup as the current page setup for another layout, the values are applied to the layout. If you make changes to the original unnamed page setup, those changes do not carry over to the layout to which the unnamed page setup was applied.

- 3** Click Modify.
- 4** In the Page Setup dialog box, make the required changes. Click OK.
- 5** In the Page Setup Manager, click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab. Click Page Setup Manager

To import named page setups from another drawing

- 1 Click File menu ► Page Setup Manager.
- 2 In the Page Setup Manager, click Import.
- 3 In the Select Page Setup From File dialog box, select a drawing file from which you want to import one or more named page setups. Click Import.
- 4 In the Import Page Setups dialog box, select one or more page setups to import. Click OK.

If a page setup with the same name already exists in the drawing, you can redefine the settings of the existing one with the settings of the imported page setup, or you can cancel the operation.

The imported page setups are displayed in the Page Setup Manager in the list of page setups.

NOTE You can import both model space and layout page setups at the same time. However, an imported model space page setup is shown in the Page Setup Manager only if the model tab was current when you opened the Page Setup Manager. Similarly, an imported layout page setup is shown in the Page Setup Manager only if a layout tab was current when you opened the Page Setup Manager.

- 5 In the Page Setup Manager, click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab. Click Page Setup Manager

To delete a named page setup

- 1 Click File menu ► Page Setup Manager.

- 2 In the Page Setup Manager, under Page Setups, right-click the named page setup you want to delete. Click Delete.

NOTE A page setup that is not named is shown in the Page Setup Manager with an asterisk on either side of its layout name. An unnamed page setup does not have the same functionality as a named page setup. For example, if you set an unnamed page setup as the current page setup for another layout, the values are applied to the layout. If you make changes to the original unnamed page setup, those changes do not carry over to the layout to which the unnamed page setup was applied.

- 3 Click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab. Click Page Setup Manager

To rename a page setup

- 1 Click File menu ► Page Setup Manager.
- 2 In the Page Setup Manager, under Page Setups, right-click the named page setup you want to rename. Click Rename.

NOTE A page setup that is not named is shown in the Page Setup Manager with an asterisk on either side of its layout name. An unnamed page setup does not have the same functionality as a named page setup. For example, if you set an unnamed page setup as the current page setup for another layout, the values are applied to the layout. If you make changes to the original unnamed page setup, those changes do not carry over to the layout to which the unnamed page setup was applied.

- 3 Enter a new name for the page setup.
- 4 Click Close.

Layouts toolbar



Command line: PAGESETUP

Shortcut menu: Right-click a layout tab. Click Page Setup Manager

Use Named Page Setups with Sheet Sets

You can use named page setups to specify the same output options for all the sheets in a sheet set.

You can create a sheet from scratch using the Sheet Set Manager, and then apply a named page setup to the layout.

You can also apply named page setups that are stored in the sheet set's page setup overrides DWT file to a single sheet or to an entire sheet set for a one-time publish operation.

See also:

["Create and Use Named Page Setups" on page 966](#)

To apply a named page setup to a sheet created from scratch

- 1 Click File menu ➤ Open Sheet Set.
- 2 In the Open Sheet Set dialog box, select a sheet set. Click Open.
- 3 In the Sheet Set Manager, right click in the Sheets area. Click New Sheet.
- 4 In the New Sheet dialog box, enter a number and sheet title. Click OK.
- 5 In the Sheet Set Manager, under Sheets, double-click the new sheet.
- 6 Click File menu ➤ Page Setup Manager.
- 7 In the Page Setup Manager, under Page Setups, select a named page setup. Click Set Current.
Named page setups are not enclosed in asterisks.
- 8 Click Close.

Standard toolbar



Command line: SHEETSET

To apply a named page setup as an override when publishing sheets

- 1 Click File menu ➤ Open Sheet Set.
- 2 In the Open Sheet Set dialog box, select a sheet set. Click Open.

- 3** In the Sheet Set Manager, in the Sheets area, select a sheet set, a subset, or a sheet to publish.
- 4** At the top of the Sheet Set Manager, click the Publish button.
- 5** On the shortcut menu, place your cursor over Publish Using Page Setup Override.
- 6** On the submenu, click the named page setup you want to use.

Standard toolbar



Command line: SHEETSET

30

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.

In this chapter

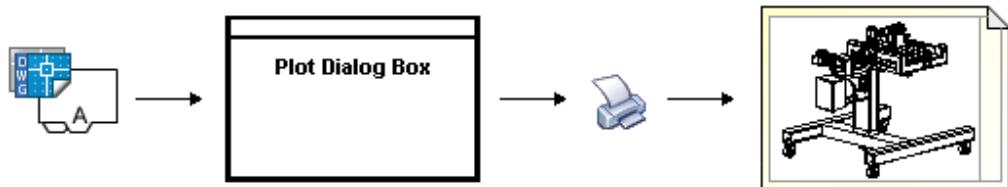
- Quick Start to Plotting
- Overview of Plotting
- Use a Page Setup to Specify Plot Settings
- Select a Printer or Plotter
- Specify the Area to Plot
- Set Paper Size
- Position the Drawing on the Paper
- Control How Objects Are Plotted
- Preview a Plot
- Plot Files to Other Formats

Quick Start to Plotting

To print a single layout or part of a drawing, use the Plot dialog box.

Use a named page setup or change the settings in the Plot dialog box to define the output of your drawing.

To output more than one drawing, use the Publish dialog box.



See also:

["Quick Start to Publishing" on page 1028](#)

Overview of Plotting

Understanding terms and concepts that relate to plotting makes your first plotting experience in the program easier.

Plotter Manager

The Plotter Manager is a window that lists plotter configuration (PC3) files for every nonsystem printer that you install. Plotter configuration files can also be created for Windows® system printers if you want to use default properties different from those used by Windows. Plotter configuration settings specify port information, raster and vector graphics quality, paper sizes, and custom properties that depend on the plotter type.

The Plotter Manager contains the Add-a-Plotter wizard, which is the primary tool for creating plotter configurations. The Add-a-Plotter wizard prompts you for information about the plotter you want to set up.

Layouts

A layout represents a plotted page. You can create as many layouts as you need. Each layout is saved on its own layout tab and can be associated with a different page setup.

Elements that appear only on a plotted page, such as title blocks and notes, are drawn in paper space in a layout. The objects in the drawing are created

in model space on the Model tab. To view these objects in the layout, you create layout viewports.

Page Setups

When you create a layout, you specify a plotter and settings such as page size and plot orientation. These settings are saved in a page setup. You can control these settings for layouts and for the Model tab using the Page Setup Manager. You can name and save page setups for use with other layouts.

If you don't specify all the settings in the Page Setup dialog box when you create a layout, you can set up the page just before you plot. Or you can override a page setup at plot time. You can use the new page setup temporarily for the current plot, or you can save the new page setup.

Plot Styles

A plot style controls how an object or layer is plotted by determining plotted properties such as linewidth, color, and fill style. Plot style tables collect groups of plot styles. The Plot Style Manager is a window that shows all the plot style tables available.

There are two plot style types: color-dependent and named. A drawing can use only one type of plot style table. You can convert a plot style table from one type to the other. You can also change the type of plot style table a drawing uses once it has been set.

For *color-dependent plot style tables*, an object's color determines how it is plotted. These plot style table files have .ctb extensions. You cannot assign color-dependent plot styles directly to objects. Instead, to control how an object is plotted, you change its color. For example, all objects assigned the color red in a drawing are plotted the same way.

Named plot style tables use plot styles that are assigned directly to objects and layers. These plot style table files have .stb extensions. Using them enables each object in a drawing to be plotted differently, independent of its color.

Plot Stamps

A plot stamp is a line of text that is added to your plot. You can specify where this text is located on the plot in the Plot Stamp dialog box. Turn this option on to add specified plot stamp information—including drawing name, layout name, date and time, and so on—to a drawing that is plotted to any device. You can choose to record the plot stamp information to a log file instead of plotting it, or in addition to plotting it.

NOTE A drawing file or drawing template file that was created with an educational version will always be plotted with the following plot stamp: PRODUCED BY AN AUTODESK EDUCATIONAL PRODUCT. Blocks and xrefs created with an educational

version and used in a commercial version will also result in the educational plot stamp being plotted.

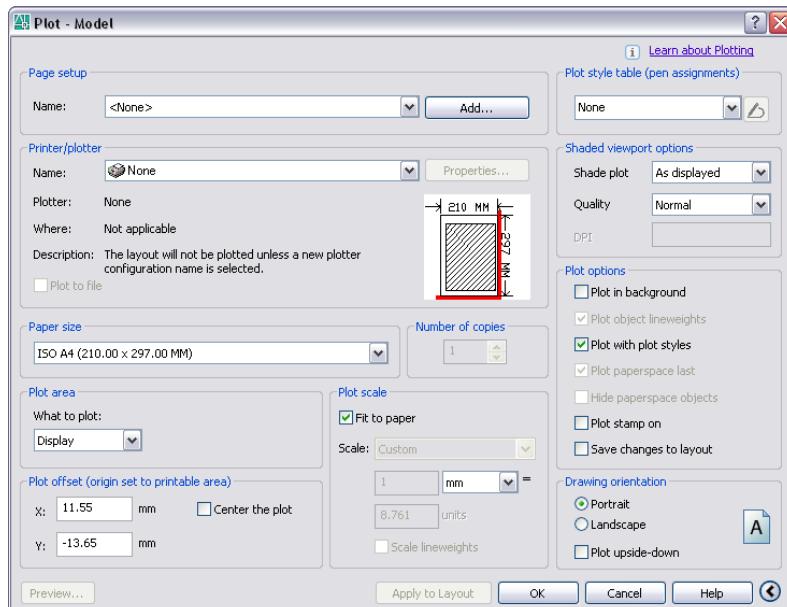
See also:

[“Create Multiple-View Drawing Layouts \(Paper Space\)” on page 209](#)

[“To view the custom properties Help” in the *Driver and Peripheral Guide*](#)

To plot a drawing

- 1 Click File menu ► Plot.



- 2 In the Plot dialog box, under Printer/Plotter, select a plotter from the Name list.
- 3 Under Paper Size, select a paper size from the Paper Size box.
- 4 (Optional) Under Number of Copies, enter the number of copies to plot.
- 5 Under Plot Area, specify the portion of your drawing to plot.
- 6 Under Plot Scale, select a scale from the Scale box.
- 7 For more options, click the More Options button.



- 8 (Optional) Under Plot Style Table (Pen Assignments), select a plot style table from the Name box.
- 9 (Optional) Under Shaded Viewport Options and Plot Options, select any appropriate settings.

NOTE Plot stamping happens at plot time and is not saved with the drawing.

10 Under Drawing Orientation, select an orientation.

11 Click OK.

Standard toolbar



Command line: PLOT

Shortcut menu: Right-click the Model tab or a layout tab and click Plot.

To plot on the command line instead of in a dialog box, use -PLOT.

To turn background plotting on or off

- 1 Click Tools menu ▶ Options.
- 2 In the Options dialog box, Plot and Publish tab, under Background Processing Options, select or clear the Enable Background Plot When Plotting option.
- 3 Click OK.

NOTE When you plot in the background, you can return immediately to your drawing. While a plot job is being processed in the background, you can check the status of your job by placing your cursor over the plotter icon in the status tray. You can also view details about all completed jobs from the current program session.

Command line: OPTIONS

To check the status of a plot job currently processing in the background

- In the status tray, place your mouse over the plotter icon. The tooltip displays the status of the plot job.

Status tray



To cancel part or all of a plot job that is processing in the background

- 1 In the status tray, right-click the plotter icon. Click Cancel Sheet <sheetname> or Cancel Entire Job.

Status tray



To view details about jobs you have plotted

- 1 Do one of the following:
 - Click File menu > View Plot and Publish Details.
 - In the status tray, click the plotter icon.
- 2 In the Plot and Publish Details dialog box, view details about plotted jobs.

Command line: VIEWPLOTDETAILS

Shortcut menu: In the status tray, right-click the plotter icon. Click View Plot and Publish Details.

Use a Page Setup to Specify Plot Settings

You can use a page setup to specify the settings for your plot job. When you select a page setup in the Plot dialog box, the settings from the page setup are added to the Plot dialog box. You can choose to plot with those settings, or change any of the settings individually and then plot.

Any settings you specify in the Plot dialog box can be saved as a new named page setup by clicking the Add button in the Page Setup area.

Any settings specified in the Plot dialog box, whether you've applied a page setup from the Page Setup list, or changed the settings individually, can be saved to the layout for use the next time you plot.

To plot a drawing using the settings specified in a page setup

- 1 Click File menu > Plot.

- 2** In the Plot dialog box, under Page Setups, select a page setup from the list.
- 3** (Optional) Change any of the individual settings in the Plot dialog box.
- 4** To save these settings to the layout, click Apply to Layout.
- 5** Click Plot.

Standard toolbar



Command line: PLOT

To save plot settings as a new named page setup

- 1** Click File menu ► Plot.
- 2** In the Plot dialog box, change the required settings.
- 3** Under Page Setups, click Add.
- 4** In the Add Page Setup dialog box, enter a name for the new named page setup. Click OK.

Standard toolbar



Command line: PLOT

To save plot settings to the layout

- 1** With a layout tab selected, click File menu ► Plot.
- 2** In the Plot dialog box, under Page Setups, select a page setup or specify settings individually.
- 3** Click Apply to Layout.

Standard toolbar



Command line: PLOT

Select a Printer or Plotter

After selecting a printing or plotting device, you also can easily plot a drawing using the default settings in the Plot dialog box.

To select a printer or plotter

- 1 Click File menu ► Plot.
- 2 In the Plot dialog box, under Printer/Plotter, select a plotter from the Name list.
If you have already selected a paper size and it is not supported by the plotter you have chosen, you are warned that a paper size supported by the plotter will be used. Click OK if the warning is displayed.
- 3 Once you have selected a plotter, you can continue to select a paper size, or if the paper size is correct, click OK to plot the drawing.

Standard toolbar



Command line: PLOT

Specify the Area to Plot

When plotting a drawing, you must specify the area of the drawing to plot. The Plot dialog box provides the following options under Plot Area.

- **Layout or Limits.** When plotting a layout, plots everything within the printable area of the specified paper size, with the origin calculated from 0,0 in the layout. When plotting the Model tab, plots the entire drawing area defined by the grid limits. If the current viewport does not display a plan view, this option has the same effect as the Extents option.
- **Extents.** Plots the portion of the current space of the drawing that contains objects. All geometry in the current space is plotted. The drawing might be regenerated to recalculate the extents before plotting.
- **Display.** Plots the view in the current viewport in the Model tab or the current paper space view in a layout tab.
- **View.** Plots a view saved previously with the VIEW command. You can select a named view from the list provided. If there are no saved views in the drawing, this option is unavailable.

■ **Window.** Plots any portion of the drawing you specify. Click the Window button to use a pointing device to specify opposite corners of the area to be plotted, or enter coordinate values.

To set the drawing area as you plot

- 1 Click File menu > Plot.
- 2 In the Plot dialog box, under Plot Area, specify the portion of your drawing that you want to plot.
- 3 Change other settings as needed. Click OK to plot the drawing.

Standard toolbar



Command line: PLOT

Set Paper Size

In the Plot dialog box, select the paper size that you want to use.

If you plot from a layout, you may have already specified a paper size in the Page Setup dialog box. However, if you plot from the Model tab, you need to specify a paper size when you plot. In the Plot dialog box, select the paper size that you want to use. The list of paper sizes depends on the printer or plotter that you have selected in either the Plot or Page Setup dialog box. The list of available plotters includes all those that are currently configured for use with Windows and those for which you have installed nonsystem drivers.

You can also set the default page size used to create new layouts for most plotters by editing the PC3 file associated with that plotter. For Windows system printers, you can use this technique to specify different default page sizes for Windows and for this program.

NOTE If the PAPERUPDATE system variable is set to 0, you are prompted if the layout's existing paper size is not supported by the plotter you have selected. If the PAPERUPDATE system variable is set to 1, the paper size is automatically updated to reflect the default paper size of the selected plotter.

Use a Custom Paper Size

If you need to specify a paper size that is not listed in either the Plot dialog box or the Page Setup dialog box, you can add a custom paper size for a

nonsystem plotter using the Plotter Configuration Editor. Typically, you cannot add a custom paper size to Windows system printers because the allowable page sizes and printable areas are determined by the manufacturer. However, you can modify the printable area for paper sizes associated with a Windows system printer.

To select a paper size for the current plot

- 1 Click File menu ► Plot.
- 2 In the Plot dialog box, under Printer/Plotter, select a plotter in the Name box.
- 3 Under Paper Size, select a paper size from the list.

The paper sizes that are listed depend on the plotter you selected.

Standard toolbar



Command line: PLOT

To set the default paper size for a layout

- 1 Click File menu ► Page Setup Manager.
- 2 In Page Setup Manager, Page Setups area, the layout for which you want to set the paper size should be selected. If not, select the layout. Click Modify.
- 3 In the Page Setup dialog box, under Paper Size, select a paper size from the list. Click OK.

The paper sizes that are listed depend on the plotter specified in the page setup.
- 4 In Page Setup Manager, click Close.

The layout reflects the changes.

Layouts toolbar



Command line: PAGESETUP

To set the default paper size for a plotter

- 1 Click File menu ▶ Plot.
- 2 In the Plot dialog box, under Printer/Plotter, select a plotter from the Name list.
- 3 Click Properties.
The Plotter Configuration Editor is displayed.
- 4 To specify a default paper size, use one of the following methods:
 - For a nonsystem plotter, under Media, select Source and Size.
 - For a system printer, in the tree view, select Custom Properties. Then under Access Custom Dialog, select Custom Properties.
- 5 Select the appropriate paper size.
- 6 Click OK to close each dialog box.

NOTE The available paper sizes depend on the printer or plotter. Some plotter manufacturers may control paper size differently in the Plotter Configuration Editor.

Layouts toolbar



Command line: PAGESETUP

To create or edit a custom paper size for a nonsystem printer

- 1 Click File menu ▶ Plotter Manager.
- 2 In the Plotter Manager, double-click the PC3 file whose configuration you want to change.
- 3 In the Plotter Configuration Editor, Device and Document Settings tab, under the User Defined Paper Sizes & Calibration, select Custom Paper Sizes.
- 4 Set a new paper size using one of the following methods:
 - To add a custom paper size, click Add and follow the steps in the Custom Paper Size wizard. You specify the size of the paper, the printable area, and a name for the new paper size.

- To edit an existing paper size, under Custom Paper Sizes, select the paper size and click Edit. The Custom Paper Size wizard opens. Change any of the paper size settings.

5 Click OK.

The new or edited paper size is available in both the Plot and Page Setup dialog boxes when that PC3 file is selected.

NOTE Creating a custom paper size for a nonsystem driver attaches a plot model parameter (PMP) file to the plotter configuration (PC3) file. The PMP file contains custom plotter calibration and custom paper size information. By default, PMP files are stored in the *Drv* folder.

Command line: PLOTTERMANAGER

Position the Drawing on the Paper

There are several ways to position a drawing on the paper. You can specify the printable area, set the position of the plot, and set the drawing orientation.

Specify the Printable Area

The printable area is displayed by a dashed border in a layout. The plotter and paper size you select determine the printable area.

WARNING If you set your plotter to use paper-saving features such as plotting inked area or nesting, your plotter will probably not use the printable area and plot offset specifications.

If your plotter reports an incorrect printable area for your paper size, you can adjust the printable area in the Modify Standard Paper Sizes area under the Modify Standard Paper Sizes (Printable Area) option on the Device and Document Settings tab in the Plotter Configuration Editor.

NOTE The Modify Standard Paper Sizes option is not a margins feature. Specify where your drawing is plotted on the page in the Plot Offset area in the Plot dialog box.

Set the Position of the Plot

The printable area of a drawing sheet is defined by the selected output device and is represented by the dashed line in a layout. When you change to another output device, the printable area may change.

The settings in the Plot Offset area of the Plot dialog box specify an offset of the plot area relative to the lower-left corner (the origin) of the printable area or the edge of the paper, depending on the setting made in the Specify Plot Offset Relative To option (Options dialog box, Plot and Publish tab). The Plot Offset area of the Plot dialog box displays the specified plot offset option in parentheses.

You can offset the drawing on the paper by entering positive or negative values in the X and Y offset boxes. However, this may result in the plot area being clipped. If the Plot Area is not set to Layout (Extents, Display, View, or Window), you can also select the Center the Plot option.

Set Drawing Orientation

The drawing orientation determines whether the position of the plotted drawing is landscape (the longer edge of the drawing is horizontal) or portrait (the longer edge of the drawing is vertical). This is based on the size of paper selected. You can also choose to plot upside down.

Control How Objects Are Plotted

You can control how objects are plotted by setting the plot scale, by using plot styles and plot style tables, and by setting an object's layer properties.

Set Plot Scale

When you specify a scale to output your drawing, you can choose from a list of real-world scales, enter your own scale, or select Fit to Paper to scale the drawing to fit onto the selected paper size.

Usually, you draw objects at their actual size. That is, you decide how to interpret the size of a unit (an inch, a millimeter, a meter) and draw on a 1:1 scale. For example, if your unit of measurement is millimeters, then every unit in your drawing represents a millimeter. When you plot the drawing, you either specify a precise scale or fit the image to the paper.

Most final drawings are plotted at a precise scale. The method used to set the plot scale depends on whether you plot the Model tab or a layout:

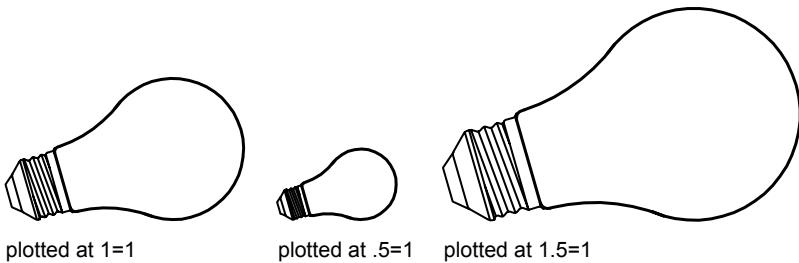
- On the Model tab, you can establish the scale in the Plot dialog box. This scale represents a ratio of plotted units to the world-size units you used to draw the model.
- In a layout, you work with two scales. The first affects the overall layout of the drawing, which usually is scaled 1:1, based on the paper size. The second is the scale of the model itself, which is displayed in layout viewports. The scale in each of these viewports represents a ratio of the paper size to the size of the model in the viewport.

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

Set a Specific Scale

When you plot, the paper size you select determines the unit type, inches or millimeters. For example, if the paper size is in mm, entering **1** under mm and **10** under Units produces a plotted drawing in which each plotted millimeter represents 10 actual millimeters.

The illustrations show a light bulb plotted at three different scales.



Scale the Drawing to Fit the Page

When you review drafts, a precise scale is not always important. You can use the Fit to Paper option to plot the view at the largest possible size that fits the paper. The height or width of the drawing is fit to the corresponding height or width of the paper.

When you plot a perspective view from model space, the view is scaled to fit the paper even when you enter a scale.

When you select the Fit to Paper option, the text boxes change to reflect the ratio of plotted units to drawing units. This scale is updated whenever you change the paper size, plotter, plot origin, orientation, or size of the plotted area in the Plot dialog box.

NOTE This option is not available when the Plot Area is set to Layout.

To plot using a real-world scale

- 1 Click File menu ► Plot.
- 2 In the Plot dialog box, under Plot Scale, select a scale from the Scale box.
- 3 Click OK to plot the drawing.

Standard toolbar



Command line: PLOT

To plot using a custom scale

- 1 Click File menu ► Plot.
- 2 In the Plot dialog box, under Plot Scale, enter a custom scale. The scale requires two values, the number of plotted units (inches or mm) per the number of drawing units. The type of unit is determined by the paper size, but you can change it in the list box.

If you enter a custom scale, Custom is automatically selected in the Scale box, even if the scale you enter is the same as a standard scale in the list. A custom scale is the ratio between the plotted units and drawing units. For example, 1:12 and 2:24 are plotted at the same scale.

- 3 Click OK to plot the drawing.

Standard toolbar



Command line: PLOT

To scale a drawing to fit the page

- 1 Click File menu ► Plot.

- 2 In the Plot dialog box, under Plot Scale, select the Fit to Paper option. The resulting scale is automatically calculated. The ratio of plotted units to drawing units in the custom scale boxes is displayed.
- 3 Click OK to plot the drawing.

NOTE This option is not available when the Plot Area is set to Layout.

Standard toolbar



Command line: PLOT

Set Shaded Viewport Options

You can choose among several options for plotting shaded and rendered viewports. You can plot a viewport as it is displayed, in wireframe, with hidden lines removed, or as rendered.

You can specify how each viewport should be plotted, and save the plot settings with a drawing. You also can choose from a wide variety of resolutions, up to the resolution of your plotter, and save the resolution settings with a drawing.

Overview of Shaded Viewport Plotting

The options for shaded viewport plotting give you a large degree of flexibility in conveying your three-dimensional designs to others. You can convey your design intent by choosing how viewports are plotted and by specifying resolution levels.

With shaded plotting options, you can choose whether to plot a set of shaded objects as displayed or in wireframe, hidden mode, a visual style, or rendered. Shaded and rendered viewports are plot-previewed, plotted, plotted to file, and published with full shading and rendering.

You can use realistic plots in your presentations by plotting viewports as they are displayed on the screen or otherwise.

Shaded viewport plotting options apply to all objects in viewports and model space. If you use the Shaded or Rendered options plot style tables included in the page setup do not affect plots. If you use the Render option, two-dimensional wireframe objects, such as lines, arcs, and text, are not plotted.

NOTE Shaded viewport plotting requires a raster-capable device. Most modern plotters and printers are raster-capable devices.

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.

You can select an option for your drawing either from model space or from a layout. From model space, the options are available in the Properties palette and the Plot dialog box. From a layout, after you select a viewport, the options are available from the shortcut menu and from the Properties palette.

Either way, you can save your settings with the drawing.

NOTE If you select the Rendered option, specify Render settings before plotting. If the Rendered option is used for a highly complex set of objects, the hardcopy output might contain only the viewport border.

To modify the shade plot setting of a viewport

- 1 Make sure you are on a layout tab.
- 2 Double-click the border of the viewport you want to modify.
- 3 In the Properties palette, under Misc., select Shade Plot, and then select an option for plotting.

Shortcut menu: Select the viewport and right-click in the drawing area. Click a setting from the Shade Plot options.

To modify the shade plot setting in model space

- 1 Make sure you are on the Model tab and no objects are selected.
- 2 Click File menu ▶ Plot.

- 3 In the Plot dialog box, under Shaded Viewport Options, Shade Plot drop-down list, select an option.

Command line: PLOT

Specify a Resolution Level for Shaded Plotting

You can set the resolution of shaded plots for either greater speed or higher fidelity.

After you select an appropriate plotter, you can specify the level of quality for plotted output. The quality level determines the dots per inch (dpi). The dpi that corresponds to a quality level is based on the plotter you select.

The maximum dpi available is also based on the plotter you select. You can specify a custom quality level and directly change the dpi to a setting between 100 and the maximum dpi of the plotter.

The higher the fidelity, the more computer memory is used, so the longer it takes to plot. High fidelity is not necessary for all plots, and a setting between 300 and 600 dpi is generally sufficient for most plots.

To specify a resolution level for shaded plotting

- 1 Click File menu ► Plot.
- 2 In the Plot dialog box, under Shaded Viewport Options, Quality drop-down list, select an option.
If you want to enter a custom dpi, select Custom and specify a dpi in the DPI box.
- 3 Click OK.

Command line: PLOT

Set Options for Plotted Objects

In the Plot and the Page Setup dialog boxes, you can choose from options that affect how objects are plotted.

- **Shaded Viewport Plotting.** Specifies shaded plotting options: As Displayed, Wireframe, or Hidden. The effect of this setting is reflected in the plot preview, but not in the layout.
- **Plot Object Lineweights.** Specifies that lineweights assigned to objects and layers are plotted.

■ **Plot with Plot Styles.** Specifies that the drawing is plotted using plot styles. Selecting this option automatically plots linewidths. If you do not select this option, objects are plotted with their assigned properties and not with the plot style overrides.

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

■ **Plot Paper Space Last.** Specifies that objects in model space are plotted before those in paper space.

■ **Hide Paperspace Objects.** Specifies whether the Hide operation applies to objects in the paper space viewport. This option is available only from a layout tab. The effect of this setting is reflected in the plot preview, but not in the layout.

■ **Plot Stamp On.** Turns on plot stamping and places a plot stamp on a specified corner of each drawing and/or logs it to a file. Plot stamp settings are specified in the Plot Stamp dialog box, where you can specify the information you want applied to the plot stamp, such as drawing name, date and time, plot scale, and so on. To open the Plot Stamp dialog box, select Plot Stamp On in the Plot dialog box, and then click the Plot Stamp Settings button.

■ **Save Changes to Layout.** Saves changes you make in the Plot dialog box to the layout if you click OK.

To set shaded plotting options from the Model tab

- 1 Click File menu ► Plot.
- 2 In the Plot dialog box, Shaded Viewport Options area, Shade Plot drop-down list, select an option.

Standard toolbar



Command line: PLOT

To set shaded plotting options from a layout tab

- 1 Select a layout viewport.
- 2 Double-click the viewport border to display the Properties palette.
- 3 On the Properties palette, click Shade Plot.

- 4** In the Shade Plot drop-down list, select an option.

Standard toolbar

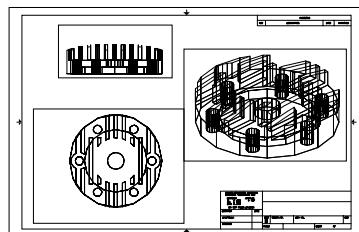


Command line: PROPERTIES

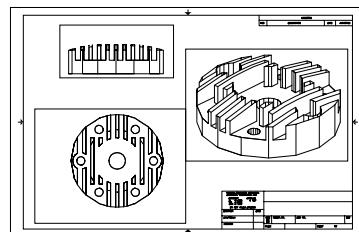
Shortcut menu: Select the viewport, right-click in the drawing area, and then click Shade Plot.

To remove hidden lines when plotting from the Model tab

- 1** Click File menu > Plot.
- 2** In the Plot dialog box, under Shaded Viewport Options area, Shade Plot drop-down list, select Hidden.
- 3** Click OK to plot the drawing.



Hidden lines not removed



Hidden lines removed

Standard toolbar

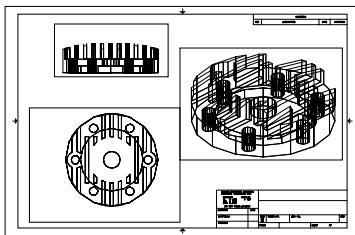


Command line: PLOT

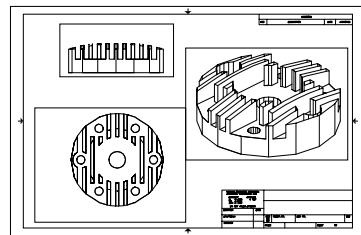
To remove hidden lines when plotting from a layout tab

- 1** Select a layout viewport.
- 2** Double-click the viewport border to display the Properties palette.

- 3 On the Properties palette, click Shade Plot.
- 4 In the Shade Plot drop-down list, select Hidden.
- 5 Click OK to plot the drawing.



Hidden lines not removed



Hidden lines removed

Standard toolbar



Command line: PROPERTIES

Shortcut menu: Select the viewport and right-click in the drawing area. Click Shade Plot.

To plot lineweights

- 1 Click File menu ► Plot.
- 2 In the Plot dialog box, under Plot Options, select Plot Object Lineweights. You can change this option only if the Plot with Plot Styles option is cleared.
- 3 Click OK to plot the drawing.

Standard toolbar



Command line: PLOT

To turn off plot styles

- 1 Click File menu ► Plot.
- 2 In the Plot dialog box, under Plot Options, clear the Plot with Plot Styles option.
- 3 Click OK to plot the drawing.

Standard toolbar



Command line: PLOT

To change the order in which objects are plotted

- 1 Click File menu ► Plot.
- 2 In the Plot dialog box, under Plot Options, select Plot Paperspace Last. Clear the Plot Paperspace Last option to plot paper space first.
- 3 Click OK to plot the drawing.

Standard toolbar



Command line: PLOT

To hide paper space objects when plotting from a layout tab

- 1 Click File menu ► Plot.
- 2 In the Plot dialog box, under Plot Options, select Hide Paperspace Objects.
- 3 Click OK to plot the drawing.

Standard toolbar



Command line: PLOT

Use Plot Styles to Control Plotted Objects

You can control many aspects of how an object is plotted by using plot styles.

Overview of Plot Styles

A plot style is an object property, similar to linetype and color. A plot style can be assigned to an object or assigned to a layer. A plot style controls an object's plotted properties, including

- Color
- Dither
- Grayscale
- Pen number
- Virtual pen
- Screening
- Linetype
- Lineweight
- Line end style
- Line join style
- Fill style

Using plot styles gives you great flexibility because you can set them to override other object properties or turn off the override as needed.

Groups of plot styles are saved in either of two types of plot style tables: color-dependent (CTB) or named (STB). Color-dependent plot style tables set style based on the color of the object. Named plot styles can be assigned to an object independent of color.

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

Choose a Type of Plot Style Table

A plot style table is a collection of plot styles assigned to a layout or the Model tab. There are two types of plot style tables: color-dependent plot style tables and named plot style tables.

Color-dependent plot style tables (CTB) use an object's color to determine characteristics such as linewidth. Every red object in a drawing is plotted the same way. While you can edit plot styles in a color-dependent plot style table, you cannot add or delete plot styles. There are 256 plot styles in a color-dependent plot style table, one for each color.

Named plot style tables (STB) contain user-defined plot styles. When you use a named plot style table, objects that have the same color may be plotted differently, based on the plot style assigned to the object. A named plot style table can contain as many or as few plot styles as required. Named plot styles can be assigned to objects or layers, just like any other property.

To set a plot style table type for new drawings

- 1 Click Tools menu ► Options.
- 2 In the Options dialog box, Plot and Publish tab, click the Plot Style Table Settings button.
- 3 In the Plot Style Table Settings dialog box, select Use Color-dependent Plot Styles or Use Named Plot Styles.
- 4 (Optional) In the Default Style Table box, select a default plot style table.
- 5 (Optional) If Use Named Plot Styles is selected, select plot styles to assign to Layer 0 and to new objects.
- 6 Click OK.

NOTE Setting plot style table types for new drawings does not affect existing drawings.

Command line: OPTIONS

Switch the Type of Plot Style Table

You can change which type of plot style table, color-dependent or named, is used in a drawing.

You can use CONVERTPSTYLES to change which type of plot style table, color-dependent or named, is used in a drawing.

When you convert a drawing from using color-dependent plot style tables to using named plot style tables, any color-dependent plot style tables attached to layouts in the drawing are removed, and named plot styles are attached in their place. If you want to use the styles defined in the color-dependent plot style tables after you convert to using named plot styles tables, you should first convert any color-dependent plot style tables to named plot style tables.

When you convert a drawing from using named plot style tables to using color-dependent plot style tables, the plot style names assigned to the objects in the drawing are lost.

In addition to changing which type of plot style table a drawing uses, you can use CONVERTCTB to convert color-dependent plot style tables to named plot style tables. You cannot, however, convert a named plot style table to a color-dependent plot style table.

To convert a drawing to use named plot styles

- 1 At the Command prompt, enter **convertpstYLES**.
- 2 Click OK when the alert box appears.
- 3 In the Select File dialog box, select a named plot style table to use for the Model tab and any layouts that use plot style tables of the same name.
- 4 Click Open.

A message is displayed to confirm that the drawing was converted.

NOTE If the Display Plot Styles option was already selected in the dialog box, you must enter **regen** on the command line to display the plot style settings.

To convert a drawing to use color-dependent plot styles

- 1 At the Command prompt, enter **convertpstYLES**.
- 2 Click OK.

A message is displayed to confirm that the drawing was converted.

To convert a color-dependent plot style table into a named plot style table

- 1 At the Command prompt, enter **convertctb**.
- 2 In the Select File dialog box, select the name of the plot style table to convert, and then click Open. By default, plot style tables are saved in the *Plot Styles* folder.
- 3 Enter the new plot style table name. Click Save.
- 4 Click OK when the alert box appears.

NOTE Be sure to use the Plot Style Table Editor to change the names of the plot styles in their new plot style table to be more meaningful *before* using the plot style table with any drawings.

Command line: CONVERTCTB

Assign Plot Style Tables to Layouts

By assigning different plot style tables to each layout in your drawing, you can control how objects in the layout are plotted. The plot style table affects both model space and paper space objects. To plot the drawing without applying plot style properties, select None from the list of plot style tables.

If you use named plot style tables, each object in the drawing either is assigned a plot style directly or inherits a plot style from its layer.

To display the effects of a plot style table in a layout, select Display Plot Styles under Plot Style Table in the Page Setup dialog box.

NOTE If you insert an xref into your current drawing, all defined plot style tables are also inserted. You can modify the appearance of your objects by editing the attached plot style tables with the Plot Style Table Editor.

To assign a plot style table to a layout

- 1 Click the Model tab or the layout tab to which you want to assign the plot style table.
- 2 Click File menu ➤ Page Setup.
- 3 In Page Setup Manager, click Modify.
- 4 Under Plot Style Table (Pen Assignments), select a plot style table from the list.
- 5 In the Question dialog box, click Yes or No to indicate whether the selection should be applied to the current tab only or to all layouts.
This option is available only for the Model tab.
- 6 To preview the effects of the plot style table in the layout, select Display Plot Styles.
This option is available only for layouts.
- 7 Click OK.
- 8 In Page Setup Manager, click Close.

NOTE If the Display Plot Styles option was already selected in the dialog box, you must enter **regen** on the command line to display the plot style settings.

Layouts toolbar



Command line: PAGESETUP

To preview the effects of a plot style table in a layout

- 1 Click the layout tab in which you want to preview the effects of the plot style table.
- 2 Click File menu ▶ Page Setup.
- 3 In Page Setup Manager, click Modify.
- 4 In the Page Setup dialog box, under Plot Style Table (Pen Assignments), select the Display Plot Styles option.
- 5 Click OK.
- 6 In Page Setup Manager, click Close.

The effects of the plot style table are previewed in the layout.

Layouts toolbar



Command line: PAGESETUP

Manage Plot Style Tables

You can use the Plot Style Manager to add, delete, rename, copy, and edit plot style tables.

Color-dependent (CTB) and named (STB) plot style tables are stored in the *Plot Styles* folder by default. This folder is also known as the Plot Style Manager.

You can use the Plot Style Manager to add, delete, rename, copy, and edit plot style tables. The Plot Style Manager lists all of the available plot style tables.

To create a plot style table

- 1 Click Tools menu ▶ Wizards ▶ Add Plot Style Table.

- 2** Read the first page. Click Next.
- 3** On the Begin page, you can choose to use an configuration file (CFG) or plotter configuration file (PCP or PC2) to import pen settings, base the new plot style table on an existing plot style table, or start from scratch. If you use an existing plot style table, the new plot style table uses the same type of table as the original. Click Next.
- 4** On the Pick Plot Style Table page, select Color-Dependent Plot Style Table or Named Plot Style Table.
- 5** If you are importing pen settings from a PCP, PC2, or CFG file, or if you are basing the new plot style table on an existing plot style table, specify the file on the Browse File Name page. If you use a CFG file, you may need to select which plotter's configuration to import. Click Next.
- 6** On the File Name page, enter a name for the new plot style table. Click Next.
- 7** On the Finish page, you can edit the new plot style table by choosing Plot Style Table Editor. You can assign the new plot style table so that it can be used in all drawings.
- 8** Click Finish.

The new plot style table is available in both the Plot and the Page Setup dialog boxes for all drawings using color-dependent plot style tables.

To rename a plot style table

- 1** Click File menu ► Plot Style Manager.
- 2** Right-click the plot style whose name you want to change. Click Rename.
- 3** Enter the new file name. Be sure to add the same file extension (.ctb or .stb). When finished, press ENTER.

The renamed plot style table is available in the Plot and Page Setup dialog boxes for all drawings using that type of plot style table.

Command line: STYLESMANAGER

To change a plot style table description

- 1** Click File menu ► Plot Style Manager.
- 2** Double-click the plot style table file whose description you want to change.
- 3** In the Plot Style Table Editor, General tab, enter the new description for the plot style table.

- 4 Click Save & Close.

Command line: STYLESMANAGER

To edit plot styles in a plot style table

- 1 Click File Menu ▶ Plot Style Manager.
- 2 Double-click the plot style table you want to change.
- 3 In the Plot Style Table Editor, Form View tab, under Plot Styles, select a plot style and edit the settings.
- 4 Click Save & Close.

Command line: STYLESMANAGER

Use Color-Dependent Plot Style Tables

When a drawing uses color-dependent plot style tables, you cannot assign a plot style to individual objects or layers. Instead, to assign plot style properties to an object, you change the color of the object or layer.

You can assign color-dependent plot style tables to layouts. You can use several predefined color-dependent plot style tables, edit existing plot style tables, or create your own.

Color-dependent plot style tables are stored in the *Plot Styles* folder and have a *.ctb* extension.

Use Predefined Color-Dependent Plot Style Tables

Several color-dependent plot style tables are installed in the *Plot Styles* folder, also known as the Plot Style Manager.

Table	Description
<i>acad.ctb</i>	Default plot style table
<i>fillPatterns.ctb</i>	Sets first 9 colors to use first 9 fill patterns, all others to use object's fill
<i>grayscale.ctb</i>	Converts all colors to grayscale when plotted
<i>monochrome.ctb</i>	Plots all colors as black
None	Applies no plot style table
<i>screening 100%.ctb</i>	Uses 100% ink for all colors

Table	Description
<i>screening 75%.ctb</i>	Uses 75% ink for all colors
<i>screening 50%.ctb</i>	Uses 50% ink for all colors
<i>screening 25%.ctb</i>	Uses 25% ink for all colors

NOTE You can assign a color-dependent plot style table to a layout only if the drawing has been set to use color-dependent plot style tables.

See also:

["Assign Plot Style Tables to Layouts" on page 998](#)

Use Named Plot Style Tables

You can only create, delete, and apply plot styles in a named plot style table. You can define as many or as few plot styles as you need in a drawing.

Use Named Plot Styles

Named plot styles are assigned to objects and layers in the same way that linetype and color are assigned to objects.

An object whose plot style is set to BYLAYER inherits the plot style assigned to its layer.

Use the Properties palette to change an object's plot style and the Layer Properties Manager to change the plot style for a layer.

Because different plot style tables can be assigned to each layout and a named plot style table can contain any number of plot styles, an object or layer may have a plot style assigned to it that is not in every plot style table. In this case, the plot style as missing in the Select Plot Style dialog box; the object's default plotting properties are used. For example, named plot style table Style1 contains plot styles A and B. Named plot style table Style2 contains plot styles B and C. In a layout that uses Style1, any objects that use plot style C are listed as having a missing plot style. Objects that are assigned plot style C in this layout are plotted using their default settings.

To change an object's plot style

You can change an object's plot style only if the drawing uses named plot style tables. If the drawing uses color-dependent plot style tables, change the object's color to alter its plotted appearance.

- 1 Select one or more objects whose plot style you want to change.

- 2** Right-click in the drawing area. Click Properties.
- 3** In the Properties palette, select a plot style from the list of available plot styles. Click the column next to Plot Style.

The plot styles listed are those already in use by objects and in the plot style table attached to the current layout.
- 4** To select a plot style from a different plot style table, select Other. In the Select Plot Style dialog box, you can attach a different plot style table to the current layout and select a plot style from that plot style table.
- 5** To edit the current plot style table, click Editor.
- 6** Click OK when finished.

Any changes made in the Properties palette are immediate. If you change the plot style table attached to the current layout, both model space and paper space objects are affected.

Standard toolbar



Command line: PROPERTIES

To change a layer's plot style

- 1** Click Format menu ► Layer.
- 2** In the Layer Properties Manager, select the layer whose plot style you want to change.
- 3** Click the current plot style in the Plot Style column. Select the plot style you want to use.
- 4** To select a plot style from a different plot style table, select an active plot style table from the Active Plot Style Tables list.

The list of plot styles changes to those in the selected plot style table.
- 5** To edit the selected plot style table, click Editor. Change settings as necessary and click Save & Close.
- 6** Click OK.

You can change a layer's plot style only if the drawing uses named plot style tables. If the drawing uses color-dependent plot style tables, change the layer's color to alter the plotted appearance of objects on the layer.

NOTE An object's plot style property can be set to BYLAYER to inherit the plot style of its layer.

Layers toolbar



Command line: LAYER

To set the current plot style

- 1 Click Format menu ► Plotstyle.
- 2 In the Current Plot Style dialog box, select a plot style from the list.
The plot styles displayed are those available in the current plot style table.
- 3 To select a plot style from a different plot style table, under Active Plot Style Table, select a plot style table.
The list of plot styles changes to those in the selected plot style table.
- 4 To edit the selected plot style table, click Editor. Change settings as necessary. Click Save & Close.
- 5 Click OK.

The plot style that you made current is used for any new objects created in the drawing.

You can change the current plot style only if the drawing uses named plot style tables. If the drawing uses color-dependent plot style tables, change the color to alter the plotted appearance of objects and layers.

NOTE The current plot style property can be set to BYLAYER to inherit the plot style of the current layer.

Command line: PLOTSTYLE

Manage Named Plot Styles

You can add, delete, rename, and copy plot styles in a named plot style table using the Plot Style Table Editor.

The Plot Style Table Editor is also used to change plot style settings for both named and color-dependent plot style tables.

NOTE You cannot delete or edit the NORMAL plot style. Also, you cannot add, delete, copy, or rename plot styles in a named plot style table if a color mapping table has been attached to the plot style table. A color mapping table associates every plot style with an ACI color.

See also:

[“Change Plot Style Settings” on page 1008](#)

To create a named plot style

- 1 Click File menu ► Plot Style Manager.
- 2 Double-click the STB file to which you want to add a plot style.
- 3 (Optional) If you want to position the plot style in the list, on the Form View tab, select the plot style that should precede the new plot style.
- 4 On the Form View or Table View tab, click Add Style.
- 5 In the Add Plot Style dialog box, enter the name of the plot style.
- 6 Click Save & Close.

NOTE You cannot edit the NORMAL plot style.

Command line: STYLESMANAGER

To copy a named plot style

- 1 Click File menu ► Plot Style Manager.
- 2 Double-click the STB file you want to edit.
- 3 In the Plot Style Table Editor, Form View tab, right-click the plot style that you want to copy. Click Copy.
- 4 Right-click a plot style. Click Paste. If you want to position the new plot style in the list, right-click the plot style that should precede the new plot style.
- 5 In the Add Plot Style dialog box, enter the name of the plot style.
- 6 Click Save & Close.

NOTE You cannot copy plot styles if the plot style table uses a color mapping table. You cannot copy plot styles in a color-dependent plot style table.

Command line: STYLESMANAGER

To change a plot style's description

- 1 Click File menu ► Plot Style Manager.

- 2** Double-click the plot style table that contains the plot style whose description you want to change.
- 3** In the Plot Style Table Editor, Form View tab, select the plot style whose description you want to change.
- 4** Under Description, change the plot style's description.
You can select additional plot styles and modify their descriptions or settings.
- 5** Click Save & Close.

NOTE You cannot delete or edit the NORMAL plot style.

Command line: STYLESMANAGER

To rename a named plot style

- 1** Click File menu ► Plot Style Manager.
- 2** Double-click the STB file that contains the plot style you want to rename.
- 3** In the Plot Style Table Editor, Form View tab, right-click the plot style whose name you want to change. Click Rename.
- 4** Enter the new name for the plot style.
- 5** Click Save & Close.

NOTE You cannot rename the NORMAL style. You cannot rename plot styles in a color-dependent plot style table.

Command line: STYLESMANAGER

To delete a named plot style

- 1** Click File menu ► Plot Style Manager.
- 2** Double-click the STB file you want to edit.
- 3** In the Plot Style Table Editor, Form View tab, select the plot style that you want to delete from the list of plot styles.
- 4** Click Delete Style.
- 5** Click Save & Close.

NOTE You cannot delete or edit the NORMAL plot style. You cannot delete plot styles if the plot style table uses a color mapping table. You cannot delete plot styles from a color-dependent plot style table.

Command line: STYLESMANAGER

Use Predefined Named Plot Style Tables

One additional named plot style table is installed for you to use beyond the default plot style table. All named plot style tables have an .stb extension.

- *acad.stb*: Default plot style table
- *Monochrome.stb*: All colors plot as black
- None: No plot style table applied

NOTE Named plot style tables are available only if the drawing has been set to use named plot style tables.

See also:

["Assign Plot Style Tables to Layouts" on page 998](#)

Delete Color Mapping Tables

If you delete the mapping table, the plot style table becomes an ordinary plot style table and is no longer useful for applying plot styles to old drawings.

Named plot style tables that you create using CFG, PCP, or PC2 files have color mapping tables that are created from your previous pen mappings. Color-dependent plot style tables also have color mapping tables. Color mapping tables are used to map plot styles to colors and thus to objects of each color when opening pre-AutoCAD 2000 drawings. This enables you to simulate the way drawings were plotted in previous versions.

While the color mapping table exists, you cannot add, delete, or rename plot styles in that plot style table.

If you delete the mapping table, the plot style table becomes an ordinary plot style table and is no longer useful for applying plot styles to old drawings. It continues to be useful for new drawings.

WARNING If you delete a color mapping table, plot styles cannot automatically be assigned to objects when older drawings are opened for the first time.

To delete a color mapping table

- 1 Click File menu ► Plot Style Manager.
- 2 Double-click the plot style table that you want to modify.
- 3 In the Plot Style Table Editor, General tab, click Delete AutoCAD Release 14 Color Mapping Table.
- 4 Read the warning. Click Yes to delete the color mapping table, or click No to keep it.
- 5 Click Save & Close.

Command line: STYLESMANAGER

Change Plot Style Settings

You can modify plot styles using the Plot Style Table Editor. Changes you make to a plot style affect the objects to which that plot style is assigned.

Overview of Plot Style Settings

You can open the Plot Style Table Editor by double-clicking a CTB or STB file in the Plotter Manager. The Plot Style Table Editor displays the plot styles contained in the specified plot style table.

The General tab lists general information about the table. The Table View and Form View tabs provide two ways to modify plot style settings. In general, the Table View tab is convenient if you have a small number of plot styles. If you have a large number of plot styles, the Form View tab might be more convenient.

In a named plot style table, the NORMAL plot style represents an object's default properties (no plot style applied). You cannot modify or delete the NORMAL style.

To edit plot style settings

- 1 Click File menu ► Plot Style Manager.
- 2 Double-click the plot style table you want to modify.
- 3 In the Plot Style Table Editor, Form View tab, under Plot Styles, select the plot style you want to modify.
- 4 Under Properties, click the arrow next to the property you want to change. Select an option from the list.

- 5 Edit other properties or plot styles as needed.
- 6 Click Save & Close.

Command line: STYLESMANAGER

Set Color, Screening, Grayscale, and Dither in Plot Style Tables

You can use a plot style to assign color, screening, grayscale, and dither properties.

Assign Plot Style Colors

The default setting for plot style color is Use Object Color. With this setting, the object retains its layer or individually set color. If you assign a plot style color, the color overrides the object's color at plot time. You can specify one of 255 ACI colors, a true color, or a color book color. The plotter must be configured for True Color if you want to plot True Color plot styles.

NOTE If you use a plot style table saved in AutoCAD 2000 or later, the True Color values change to the nearest match in the current version's palette.

Use Screening

You can select a color intensity setting that determines the amount of ink placed on the paper while plotting. The valid range is 0 through 100. Selecting 0 reduces the color to white. Selecting 100 displays the color at its full intensity. Screening is effective only if your plotter is configured to plot colors or grayscale. Also, dithering must be enabled.

Use Dithering

A plotter uses dithering to approximate colors with dot patterns, giving the impression of plotting with more colors than the ink colors available in the plotter. If the plotter does not support dithering, the dithering setting is ignored.

The most common reason for turning off dithering is to avoid false line typing from dithering of thin vectors and to make dim colors more visible. When you turn off dithering, colors are mapped to the nearest color, which limits the range of colors used for plotting. Dithering is available whether you use the object's color or assign a plot style color.

NOTE Dithering disables merge control.

Convert to Grayscale

When you select Convert to Grayscale, the object's colors are converted to grayscale if the plotter supports grayscale. Light colors, such as yellow, are plotted with light gray values. Dark colors are plotted with dark gray values. If you clear Convert to Grayscale, the RGB values are used for the object's colors. Conversion to grayscale is available whether you use the object's color or assign a plot style color.

To assign a plot style color

- 1** Click File menu ► Plot Style Manager.
- 2** Right-click a CTB or STB file. Click Open.
- 3** In the Plot Style Table Editor, Table View tab, click the Color field for the plot style you want to change.
- 4** On the Color drop-down list, click the color you want to use or click Select Color to display the Select Color dialog box and do one of the following:
 - On the Index tab, click a color or enter the ACI color number (1-255) or name in the Color box. Click OK.
 - On the True Color tab, in the Color Model box, specify a color. (Enter a color value in the Color box or specify values in the Hue, Saturation, and Luminance boxes.) Click OK.
 - On the Color Books tab, in the Color Book box, select a color (use the up and down arrow and click on a color chip.) Click OK.

Command line: STYLESMANAGER

To use screening

- 1** Click File menu ► Plot Style Manager.
- 2** Double-click the plot style table that you want to modify.
- 3** In the Plot Style Table Editor, Form View tab, select the plot style you want to change.
- 4** In the Screening box, enter an intensity value between 1 and 100.
- 5** When finished, click Save & Close.

You can edit properties for multiple plot styles while in the Plot Style Table Editor.

NOTE A common practice when using screening is to set the plotted color to black for each style so that the screening will be a percentage of black.

Command line: STYLESMANAGER

To enable or disable dithering

- 1 Click File menu ▶ Plot Style Manager.
- 2 Double-click the plot style table that you want to modify.
- 3 In the Plot Style Table Editor, Form View tab, select the plot style you want to change and select Dither.
- 4 Select On or Off.
- 5 When finished, click Save & Close.

You can edit properties for multiple plot styles while in the Plot Style Table Editor.

Command line: STYLESMANAGER

To enable or disable conversion to grayscale

- 1 Click File menu ▶ Plot Style Manager.
- 2 Double-click the plot style table that you want to modify.
- 3 In the Plot Style Table Editor, Form View tab, select the plot style you want to change and select Grayscale.
- 4 Select On or Off.
- 5 When finished, click Save & Close.

You can edit properties for multiple plot styles while in the Plot Style Table Editor.

Command line: STYLESMANAGER

Assign Pen Numbers and Virtual Pens in Plot Style Tables

The pen assignments used by the plotter differ depending on whether you use a pen plotter or another kind of printer, such as a laser printer.

Assign Pens to Plot Styles

The Use Assigned Pen Number setting in the Plot Style Table Editor specifies which physical pen to use for each plot style. The physical pens in the plotter are described in the plotter's configuration (PC3) file. If you use a pen plotter, you must provide information about the color, speed, and width of each pen in the Physical Pen Configuration section of the Plotter Configuration Editor.

For example, when you specify pen information in the Plotter Configuration Editor, you might specify that pen #1 is black and 0.010 inches and pen #2 is red and 0.020 inches. In the Plot Style Table Editor you can assign pen #1 to the plot style called WATER PIPES and pen #2 to the plot style called SEWER PIPES.

You can assign a pen to a plot style by selecting from a range of 32 pen numbers in the Use Assigned Pen Number field. The default value is 1. If plot style color is set to Use Object Color, or if you are editing a plot style in a color-dependent plot style table, you cannot change the assigned pen number.

If you specify 0, the field is updated to read Automatic. The information you provided under Physical Pen Characteristics in the Plotter Configuration Editor is used to select the pen closest in color to the color of the object you are plotting.

Assign Pen Settings to Plotters Without Pens

Many plotters that do not use pens can simulate the performance of a pen plotter by using virtual pens. For many devices, you can control the virtual pens in the device with software or, by configuring them from the plotter's control panel, with hardware.

If you allow software to control the pens, the Plot Style Table values for the Lineweight, Linetype, Screening, Line End Style, Line Join Style, and Fill Style settings are effective and override the settings on the plotter's control panel.

If you turn off software control of the pen attributes (typically done on the plotter), then the software can select virtual pens but can't control linewidth, linetype, end style, join style, fill style, or color. In the program, you select hardware (virtual pen) control over software (normal) control by selecting 255 Virtual Pens in the Color Depth area of the Vector Graphics option on the Device and Document Settings tab in the Plotter Configuration Editor. Selecting any other color depth specifies software control.

In the Plot Style Table Editor under Virtual Pen #, you can specify a virtual pen number between 1 and 255. Enter **0** or **Automatic** to specify that the virtual pen assignment should be made from the ACI.

When you create a plot style table, it is important to remember that it can be used with many different plotters and that the plotter and mode determine what parts of the plot style table are enabled.

- When using a pen plotter with user assigned pens, the virtual pen number and any color assignments are ignored.
- When using a pen plotter with automatically assigned pens, pens are selected based on entity color and entity linewidth. Virtual pen numbers are ignored.
- When using a raster plotter in raster mode, the physical pen number and the virtual pen number are ignored.
- When using a raster plotter in virtual pen mode, everything except the virtual pen number is ignored.

NOTE If you use another application to process your plot files after creating them, and you modify the pen attributes, plotting without using virtual pens results in pen numbers in the plot file having no simple relationship to object colors in the program. This makes it difficult to apply additional pen attributes.

To specify a virtual pen number

- 1 Click File menu ► Plot Style Manager.
- 2 Double-click the plot style table that you want to modify.
- 3 In the Plot Style Table Editor, Form View tab, select the plot style you want to change and click Virtual Pen.
- 4 Under Virtual Pen #, enter a number between 1 and 255 or enter **0** or **Automatic** to have the program assign the ACI color of the object you are plotting to the virtual pen.
- 5 When finished, click Save & Close.

You can edit properties for multiple plot styles while in the Plot Style Table Editor.

Command line: STYLESMANAGER

Control Plotted Lineweight and Linetype

Both linewidth and linetype can be set as an object property or controlled when plotted by using a plot style. Lineweight or linetype settings in the plot style override the object's linewidth or linetype at plot time.

Assign and Display Lineweights

When you select the Lineweight field in the Plot Style Table Editor, a sample of the linewidth as well as its numeric value are displayed. The default setting for plot style linewidth is Use Object Lineweight. You can modify an existing linewidth if the one you need is not available.

To view plot style linewidths in a layout, select Display Plot Styles under Plot Style Table in the Page Setup dialog box.

Assign Linetypes

When you select the Linetype field in the Plot Style Table Editor, a list with a sample and a description of each linetype are displayed. The default setting for plot style linetype is Use Object Linetype.

Whether you choose to assign a linetype as a property of the object or as a plot style, you can set the Adaptive Adjustment option. This option adjusts the scale of the linetype to complete the linetype pattern. If you don't select Adaptive Adjustment, the line might end in the middle of a pattern. Turn off Adaptive Adjustment if linetype scale is important. Turn on Adaptive Adjustment if complete linetype patterns are more important than correct linetype scaling.

You can apply a global scale factor to non-ISO linetypes and fill patterns in plot styles.

See also:

- “Work with Linetypes” on page 309
- “Control Lineweights” on page 317

To set the plotted linewidth

- 1 Click File menu ▶ Plot Style Manager.
- 2 Double-click the plot style table that you want to modify.
- 3 In the Plot Style Table Editor, Form View tab, select the plot style you want to change.
- 4 Click the Lineweight arrow and select a linewidth from the list. To specify that the object's linewidth should be used select Use Object Lineweight.
- 5 Click Save & Close.

You can edit properties for multiple plot styles while in the Plot Style Table Editor.

Command line: STYLESMANAGER

To set the plotted linetype

- 1 Click File menu ► Plot Style Manager.
- 2 Double-click the plot style table that you want to modify.
- 3 In the Plot Style Table Editor, Form View tab, select the plot style you want to change.
- 4 Click the Linetype arrow and select a linetype from the list. To specify that the object's linetype should be used, select Use Object Linetype.
- 5 To adjust the linetype scale to show the complete pattern, on the Form View tab, select On in the Adaptive box.
- 6 Click Save & Close.

You can edit properties for multiple plot styles while in the Plot Style Table Editor.

Command line: STYLESMANAGER

To apply scaling to non-ISO linetypes and fill patterns

- 1 Click File menu ► Plot Style Manager.
- 2 Double-click the plot style table that you want to modify.
- 3 In the Plot Style Table Editor, General tab, select Apply Global Scale Factor to Non-ISO Linetypes.

This option scales linetypes and fill patterns according to a value that you specify.
- 4 In the Scale Factor box, enter a scale factor to apply.
- 5 Click Save & Close.

Command line: STYLESMANAGER

Assign Plotted Line End and Join Styles

You can set the line end and join styles for objects that have linewidth assigned, either as an object property or as a plot style override.

Assign Line End Style

The program includes the following line end style options:

- Butt

- Square
- Round
- Diamond

The default setting for Line End Style is Use Object End Style, which is rounded. Assign a line end style in a plot style to override the object's default line end style at plot time.

NOTE SHX text plots best with the Round End and Round Join styles.

Assign Line Join Style

The program includes the following line join style options:

- Miter
- Bevel
- Round
- Diamond

The default setting for Line Join Style is Use Object Join Style, which is rounded. Assign a line join style in a plot style to override the object's default line join style at plot time.

To assign a line end or line join style

- 1 Click File menu ► Plot Style Manager.
- 2 Right-click a CTB or STB file. Click Open.
- 3 In the Plot Style Table Editor, Table View tab, click the Line End Style or Line Join Style field for the plot style you want to change.
- 4 Select an option from the drop-down list.

Command line: STYLESMANAGER

Assign Plotted Fill Styles

The program includes the following fill style options when plotting wide polylines, donuts, objects hatched with a solid fill, and solids:

- Solid

- Checkerboard
- Crosshatch
- Diamonds
- Horizontal Bars
- Slant Left
- Slant Right
- Square
- Dots
- Vertical Bar

The default setting for Fill Style is Use Object Fill Style. Assign a fill style in a plot style to override the object's fill style at plot time.

You can apply a global scale factor to non-ISO linetypes and fill patterns in plot styles.

See also:

["Choose Hatch Patterns and Solid Fills" on page 780](#)

To assign a fill style

- 1 Click File menu ► Plot Style Manager.
- 2 Right-click a CTB or STB file. Click Open.
- 3 In the Plot Style Table Editor, Table View tab, click the Fill Style field for the plot style you want to change.
- 4 Select a fill style from the drop-down list.

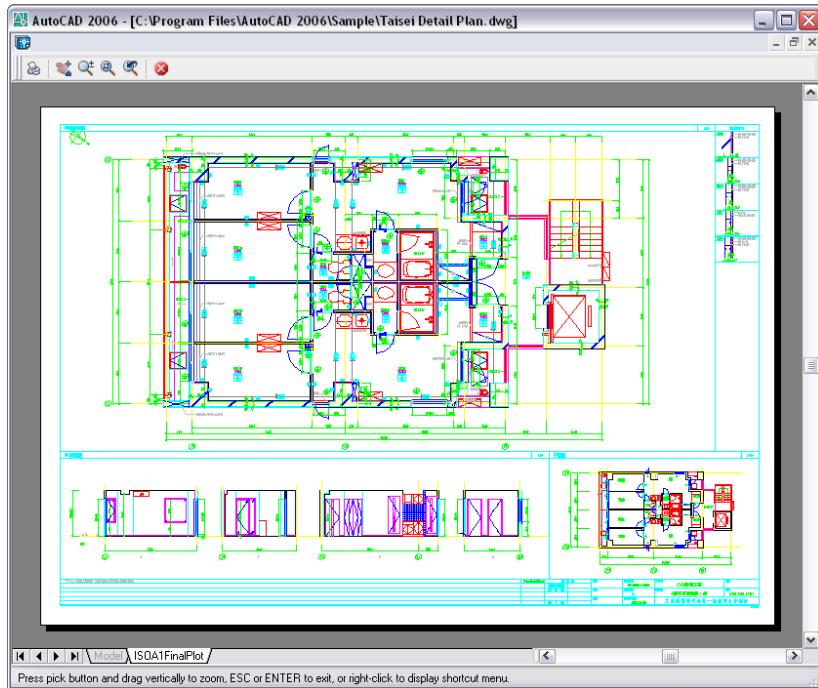
Command line: STYLESMANAGER

Preview a Plot

It is good practice to generate a preview of the plotted drawing before sending the drawing to the printer or plotter. Generating a preview saves time and material.

You can preview the drawing from the Plot dialog box. The preview shows exactly how the drawing will look when plotted, including linewidths, fill patterns, and other plot style options.

When you preview your drawing, the active toolbars and tool palettes are hidden and a temporary Preview toolbar is displayed that provides buttons to plot, pan, and zoom the drawing.



In the Plot and Page Setup dialog boxes, a thumbnail preview is also displayed, which shows the printable area and the position of the drawing on the page.

To preview a plot

- 1 Click File menu ► Plot.
- 2 In the Plot dialog box, click Preview.
A preview window opens, and the cursor changes to the real-time zoom cursor.
- 3 Right-click to display a shortcut menu with the following options: Plot, Pan, Zoom, Zoom Window, or Zoom Original (to zoom to the original preview magnification).
- 4 Press ESC to exit the preview and return to the Plot dialog box.
- 5 If necessary, make additional adjustments to the plot settings and preview the plotted drawing again.

- 6** Once the settings are correct, click OK to plot the drawing.

Standard toolbar



Command line: PLOT

Plot Files to Other Formats

You can export or plot your drawings in a number of formats, including DWF, DXF, PDF, and Windows metafile (WMF). You can also output your drawings in image formats using specially designed plotter drivers.

In each case, a nonsystem plotter driver is configured to output file information. You can control the custom properties of each nonsystem driver in the Plotter Configuration Editor. Specific help for each driver is also available by choosing Help while in the custom Properties dialog box for the individual driver (accessed through the Plotter Configuration Editor).

See also:

[“Publish Drawings” on page 1027](#)

Plot DWF Files

You can use the program to create Design Web Format (DWF) files. A DWF file is a 2D vector file that you can use to publish your drawing on the World Wide Web or an intranet network. Each DWF file can contain one or more drawing sheets.

DWF files can be opened, viewed, and plotted by anyone using Autodesk® Design Review or Autodesk® DWF™ Viewer. With Autodesk Design Review or Autodesk DWF Viewer, you can also view DWF files in Microsoft® Internet Explorer 5.01 or later. DWF files support real-time panning and zooming as well as control over the display of layers and named views.

See also:

[“Publish Drawings” on page 1027](#)

To plot a DWF file

- 1** Click File menu ► Plot.
- 2** In the Plot dialog box, under Printer/Plotter, in the Name box, select the *DWF6 ePlot.pc3* configuration from the Name list.

- 3** Select plot settings for the DWF file as needed.
- 4** Click OK
- 5** In the Browse for Plot File dialog box, select a location and enter a file name for the DWF file.
- 6** Click Save.

Standard toolbar



Command line: PLOT

Plot to DXB File Formats

DXB (drawing interchange binary) file formats are supported using the DXB nonsystem file driver. This is commonly used to “flatten” 3D drawings to 2D.

The output is compatible with the DXBIN command and with the ADI DXB driver delivered with earlier releases. The DXB driver shares these limitations of the ADI driver:

- The driver produces 16-bit integer DXB files containing only vectors.
- DXB output is monochrome; all vectors are color 7.
- Raster images and embedded OLE objects are not supported.
- The driver ignores object and plot style lineweights.

See also:

[“Configure for File Output” in the *Driver and Peripheral Guide*](#)

To create a DXB file

- 1** Make sure you have configured a plotter driver for DXB file output. (See [Configure for File Output in the *Driver and Peripheral Guide*.](#))
- 2** Click File menu ► Plot.
- 3** In the Plot dialog box, under Printer/Plotter, in the Name box, select a DXB format configuration from the list.
- 4** Select plot settings for the DXB file as needed.
- 5** Click OK.

- 6 In the Browse for Plot File dialog box, select a location and enter a file name for the DXB file.
- 7 Click Save.

Standard toolbar



Command line: PLOT

Plot to Raster File Formats

The nonsystem raster driver supports several raster file formats, including Windows BMP, CALS, TIFF, PNG, TGA, PCX, and JPEG. The raster driver is most commonly used to plot to files for desktop publishing.

All but one of the formats supported by this driver produce “dimensionless” raster files that have size in pixels but do not have size in inches or millimeters. The Dimensional CALS format is for plotters that can accept CALS files. If your plotter accepts CALS files, you must specify a real paper size and resolution. Specify the resolution in dots per inch in the Vector Graphics pane of the Plotter Configuration Editor.

By default, the raster driver plots only to files. However, you can select Show All Ports on the Ports page of the Add-a-Plotter wizard or the Ports tab in the Plotter Configuration Editor; all of the ports on your computer are then available for configuration. When configured for plotting to a port, this driver plots to a file and then copies that file to the specified port. To plot successfully, make sure that the device connected to the configured port can accept and process the file. For more information, refer to the documentation provided by the device manufacturer.

The type, size, and color depth of the raster file determine the final file size. Raster files can grow very large. Use only the pixel dimensions and color depth that you need.

You can configure the background color for raster plots in the custom Properties dialog box in the Plotter Configuration Editor. If you change the background color, any objects plotted in that color are invisible.

See also:

“Configure for File Output” in the *Driver and Peripheral Guide*

To create a raster file

- 1 Make sure you have configured a plotter driver for raster file output. (See Configure for File Output in the *Driver and Peripheral Guide*.)
- 2 Click File menu ► Plot.
- 3 In the Plot dialog box, under Printer/Plotter, in the Name box, select a raster format configuration from the list.
- 4 Select plot settings for the raster file as needed.
- 5 Click OK.
- 6 In the Browse for Plot File dialog box, select a location and enter a file name for the raster file.
- 7 Click Save.

Standard toolbar



Command line: PLOT

Plot Adobe PDF Files

Using the DWG to PDF driver, you can create Adobe® Portable Document Format (PDF) files from drawings.

The Adobe® Portable Document Format (PDF) is a standard for electronic information exchange. PDF files can be easily distributed for viewing and printing in the Adobe Reader available from the Adobe website 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 6 or later.

Use the custom Properties dialog box in the Plotter Configuration Editor to customize the output. To display this dialog box, on the Device and Document Settings tab, in the tree view, select Custom Properties. Then under Access Custom Dialog, click the Custom Properties button.

You can customize the PDF output by specifying resolution. In the Custom Properties dialog box in the Plotter Configuration Editor, you can specify the resolution for vector and raster images ranging from 150 dpi to a maximum of 4800 dpi. You can also specify custom resolutions for vector, gradient, color, and black and white output.

NOTE Although transparent objects and wipeouts are displayed correctly in the PDF viewer, they may not print with the same visual fidelity when default print settings are used. If your drawing contains transparent objects, you may need to adjust some settings in Adobe Acrobat. Set Transparency Flattening to "Print as Image" or reduce the Raster/Vector Balance in Adobe Acrobat. Refer to the Adobe documentation for more information.

See also:

"Configure for File Output" in the *Driver and Peripheral Guide*

To plot a PDF file

- 1 Click File menu ► Plot.
- 2 In the Plot dialog box, under Printer/Plotter, in the Name box, select the *DWG to PDF.pc3* configuration from the Name list.
- 3 Select plot settings for the PDF file as needed.
- 4 Click OK.
- 5 In the Browse for Plot File dialog box, select a location and enter a file name for the PDF file.
- 6 Click Save.

Standard toolbar



Command line: PLOT

To plot a PDF file in landscape orientation

- 1 Click File menu ► Plot.
- 2 In the Plot dialog box, under Printer/Plotter, in the Name box, select the *DWG to PDF.pc3* configuration from the Name list.
- 3 Under Paper Size, select a paper size that has the longer dimension listed first. For example, *ANSI A (11.00 x 8.50 Inches)*.
- 4 Click OK.
- 5 In the Browse for Plot File dialog box, select a location and enter a file name for the PDF file.
- 6 Click Save.

Standard toolbar



Command line: PLOT

Plot Adobe PostScript Files

Using the Adobe PostScript driver, you can use DWGs with an array of page layout programs and archive tools such as the Adobe Acrobat Portable Document Format (PDF).

You can use the nonsystem PostScript driver to plot drawings to PostScript printers and PostScript files. Use the PS file format for printers and the EPS file format for files. If you plot to a hardware port, PS output is automatic. If you plot to a file and plan to copy the file to a printer, configure for PS output.

Use the custom Properties dialog box in the Plotter Configuration Editor to customize the output. To display this dialog box, on the Device and Document Settings tab, in the tree view, select Custom Properties. Then under Access Custom Dialog, click the Custom Properties button.

The PostScript driver supports three types of PostScript.

- Level 1: Use for most plotters.
- Level 1.5: Use for plotters that support color images.
- Level 2: If your plotter supports Level 2 PostScript, use to produce smaller files that print more rapidly.

The Tokenize PostScript Code and Compression options in the PostScript Custom Properties dialog box reduce output file size and improve printing speed on devices that support these options. If you have problems printing, try clearing all the options. If you successfully print with no optimizations, you can try turning the options on one at a time to determine the options your printer supports.

Some desktop publishing applications only support Level 1 PostScript. If you have problems using your EPS files, try a lower PostScript level and turn off the optimizations just described.

Including a preview thumbnail in your EPS file makes the file substantially larger but allows quick preview by many applications. The WMF preview is for Windows; the EPSF preview is for Macintosh and other platforms.

NOTE Including both preview images can triple your file size.

See also:

["Export PostScript Files"](#)

["Configure for File Output" in the *Driver and Peripheral Guide*](#)

To plot a PostScript file

- 1 Make sure you have configured a plotter driver for PostScript file output. (See "Configure for File Output" in the *Driver and Peripheral Guide*.)
- 2 Click File menu ► Plot.
- 3 In the Plot dialog box, under Printer/Plotter, in the Name box, select a PostScript format configuration.
- 4 Select plot settings for the PostScript file as needed.
- 5 Click OK.
- 6 In the Browse for Plot File dialog box, select a location and enter a file name for the PostScript file.
- 7 Click Save.

Standard toolbar



Command line: PLOT

Create Plot Files

You can use any plotter configuration to create plot files that can be used with spooling software or given to service bureaus for output.

For example, HP-GL and HP-GL/2 formats are used as an intermediate format for illustration and manufacturing applications, for archive purposes, and for use with a wide range of output devices.

The HP-GL nonsystem driver supports HP-GL (Hewlett-Packard Graphics Language), a widely used pen plotter language that has vector-only capabilities. Raster objects are not supported by the HP-GL device driver.

The nonsystem HP-GL/2 driver supports a variety of HP-GL/2 pen plotters and ink-jet plotters. It is a generic HP-GL/2 driver that is not optimized for any particular manufacturer's devices. For example, it does not send PJL commands to devices, as a true Hewlett-Packard driver would. The HP-GL/2 driver supports obsolete pen plotters and newer devices made by manufacturers other than Hewlett-Packard.

See also:

"Set Device-Specific Configurations" in the *Driver and Peripheral Guide*

To create a plot file

- 1 Click File menu ▶ Plot.
- 2 In the Plot dialog box, under Printer/Plotter, in the Name box, select a plotter configuration.
- 3 If the Plot to File option is available and is not selected, select it.
- 4 Select plot settings for the plot file as needed.
- 5 Click OK.
- 6 In the Browse for Plot File dialog box, select a location and enter a file name for the plot file.
- 7 Click Save.

Standard toolbar



Command line: PLOT

3 |

Publish Drawings

Publishing provides a streamlined alternative to plotting multiple drawings. You can easily publish an entire sheet set from the Sheet Set Manager as a set of paper drawings or as a single, electronic, multisheet Design Web Format (DWF).

Publishing an electronic drawing set as a DWF file saves time and increases productivity by providing accurate, compressed representations of drawings in a file that's easy to distribute and view. DWF files can be viewed or plotted by anyone using a free, downloaded DWF file viewer, Autodesk® DWF™ Viewer.

In this chapter

- Quick Start to Publishing
- Overview of Publishing
- Create and Modify a Drawing Set for Publishing
- Create a Paper or Plot File Drawing Set
- Publish an Electronic Drawing Set
- Publish a Sheet Set
- Republish a Drawing Set
- View a Published Electronic Drawing Set
- Set Publish Options
- 3D DWF Publishing
- Configure a DWF6 Driver (Advanced)

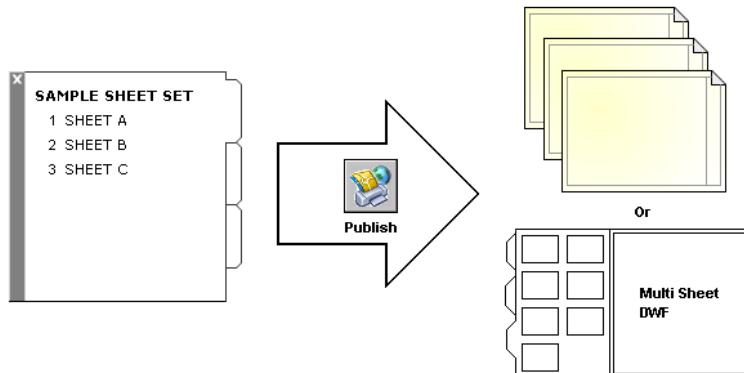
Quick Start to Publishing

Sets of drawings are the primary deliverable for most design groups. Creating a drawing set to distribute for review can be complicated and time consuming.

Using the Publish dialog box, you can easily assemble a collection of drawings and, with one click, create a paper drawing set or an electronic drawing set.

Electronic drawing sets are saved as Design Web Format™ (DWF™) files. You can view or plot DWF files using a free DWF file viewer, Autodesk® DWF™ Viewer.

To output a single drawing, use the Plot dialog box.



See also:

["Quick Start to Plotting" on page 974](#)

No procedures for this topic.

Overview of Publishing

Publishing provides an easy way to create a paper drawing set or an electronic drawing set. An electronic drawing set is the digital equivalent of a set of plotted drawings. You create an electronic drawing set by publishing drawings to a Design Web Format™ (DWF™) file.

You can publish an entire sheet set from the Sheet Set Manager. With one click, you can create an electronic drawing set by publishing the sheet set to a single, multisheet DWF.

You can create a paper drawing set by publishing the sheet set to the plotter named in each sheet's page setup.

Using the Publish dialog box, you can assemble a collection of drawings to publish and save the list as a Drawing Set Descriptions (DSD) file. You can customize this collection of drawings for a specific user, and you can add and remove sheets as a project evolves. Once you've created a list of drawing sheets in the Publish dialog box, you can publish the drawings to any of the following:

- The plotter named in each sheet's page setup (including drawings that you want to plot to file)
- A single, multisheet DWF containing both 2D and 3D content
- Multiple single-sheet DWF containing both 2D and 3D content

Using 3D DWF publishing, you can create and publish DWF files of your three-dimensional models and view them with the Autodesk DWF Viewer.

For more on publishing 3D DWF files, refer to [3D DWF Publishing](#).

Publishing an electronic drawing set as a DWF file saves time and increases productivity by providing accurate, compressed representations of drawings in a file that's easy to distribute and view. This also maintains the integrity of your original drawings.

When publishing DWF files, DWF6 files are generated, which are created in a vector-based format (except for inserted raster image content), ensuring that precision is maintained. You can view or plot DWF files using a free [DWF file viewer](#), Autodesk® DWF™ Viewer. DWF files can be distributed using email, FTP sites, project websites, or CDs.

You can specify which block-related properties and attributes you want to make available to others who are using the Autodesk DWF Viewer to view or print your published DWF files. For example, you can publish a DWF file for a plumbing contractor that contains block attribute information about the plumbing fixtures specified in your drawing data. And, from the same set of sheets, you can include only the block attribute data about light fixtures for an electrical contractor.

By default, jobs that are published are processed in the background, so that you can return immediately to your drawing. Only one job that you have published can be processed in the background at a time. While a job is being processed in the background, you can check its status by placing the cursor over the plotter icon on the right side of the status bar. You can also view details about all completed jobs that you have plotted or published from the current session.

See also:

- “[Publish, Transmit, and Archive Sheet Sets](#)” on page 261
- “[Specify Page Setup Settings](#)” on page 946

[“3D DWF Publishing” on page 1058](#)

To check the status of a published job that is being processed in the background

- Place the cursor over the plotter icon in the status tray. A tooltip displays the status of the job.

Status tray



To cancel part or all of a published job that is processing in the background

- 1 Right-click the plotter icon in the status tray. Click Cancel Sheet <sheetname> or Cancel Entire Job.

Status tray



To view details about jobs you have published

- 1 Do one of the following:
 - Click File menu ► View Plot and Publish Details.
 - In the status tray, left-click the plotter icon.
- 2 In the Plot and Publish Details dialog box, view details about published jobs.

Command line: VIEWPLOTDETAILS

Shortcut menu: In the status tray, right-click the plotter icon. Click View Plot and Publish Details.

To turn background publishing on or off

- 1 Click Tools menu ► Options.
- 2 In the Options dialog box, Plot and Publish tab, under Background Processing Options, select or clear the Enable Background Plot When Publishing option.

3 Click OK.

Command line: OPTIONS

Create and Modify a Drawing Set for Publishing

You can assemble a collection of drawing sheets to publish to a plotter, plot files, or to a DWF™ file. You can customize your drawing set for a specific user, and you can add, remove, reorder, copy, and rename sheets in a drawing set as a project evolves.

You can publish the drawing set directly to paper or to a single DWF file or multiple DWF files that can be distributed using email, FTP sites, project websites, or CD. You can save a description of a drawing set that's been assembled to publish in a Drawing Set Descriptions (DSD) file.

To create a drawing set for publishing

- 1 With a drawing open, click File menu ► Publish.

The Publish dialog box is displayed. If the Include Layouts When Adding Sheets option is selected, either in the shortcut menu or in the Publish dialog box, all the layouts in the current drawing are listed in the sheetlist.

- 2 In the Publish dialog box, you can modify the list of sheets by doing any of the following as needed:

■ **Add sheets.** To add sheets from other drawings, click the Add Sheets button (or drag drawings from the desktop). In the Select Drawings dialog box, select drawings. Click Select to add them to the list of sheets in the Publish dialog box. All of the layouts in a drawing become individual sheets in the list of drawing sheets. You can remove the individual sheets if you do not want them to become part of the drawing set. A layout must be initialized (its paper size must be defined in the page setup to any size other than 0 x 0) before it can be published.

NOTE To include all layouts when you add sheets to a drawing set, make sure the Include Layouts When Adding Sheets option is selected on the shortcut menu or in the Publish dialog box.

■ **Include model layouts.** If you include an uninitialized (the paper size is not defined in the page setup or is set to 0 x 0) model layout,

it will be marked as Uninitialized in the Status column on the sheet list. It can be plotted if you select an override page setup for it in the Publish dialog box from the page setup drop-down list under Page Setups in the sheet list.

NOTE To include the model when you add sheets to a drawing set, make sure the Include Model When Adding Sheets option is selected either on the shortcut menu or in the Publish dialog box.

- **Remove sheets.** To remove sheets from the list, select one or more sheets in the list. Click the Remove Sheets button. To remove all sheets, right-click. Click Remove All.
- **Reorder sheets.** To reorder the sheets up or down one position in the list, select a sheet. Click either the Move Sheet Up or Move Sheet Down button. Sheets in the drawing set are viewed or plotted in the order shown in the list.
- **Rename sheets.** To rename a sheet, select it in the list and right-click. Click Rename Sheet. Enter the new sheet name.
- **Change page setups.** To change the page setup for a layout, select the sheet, and in the Page Setup list select a named page setup from the list; or select one or more sheets and right-click. Click Change Page Setup. In the Page Setup list, select a page setup, or select Import to import page setups from another drawing or template. In the Import Page Setups dialog box, select a drawing with one or more page setups. Click Import. In the Page Setup list, select a named page setup.

NOTE It is important that you change the page setup for each layout depending on your desired output. Model space page setups can only be applied to model space sheets; paper space page setups can only be applied to paper space sheets.

- **Copy sheets.** To copy one or more drawing sheets, select the sheets in the list and right-click. Click Copy Selected Sheets. The copied drawing sheets are added and highlighted at the end of the sheet list. When a sheet is copied, its name is created by the addition of *-Copy(n)* at the end of the original sheet name. For example, if you create one copy of a sheet called *Plumbing*, the copied sheet is called *Plumbing-Copy(1)*. Each time you copy the same sheet, the *n* is incremented by 1. By creating copies of a sheet, you can have different page setups and other settings for the same sheet.

- 3 When your list of drawing sheets is assembled and configured the way you want for your drawing set, click the Save List button.

- 4 In the Save List As dialog box, in the File Name box, enter a name for the list. Click Save.

The drawing set list is saved as a DSD (Drawing Set Descriptions) file.

Standard toolbar



Command line: PUBLISH

To add sheets from a drawing to a drawing set for publishing

- 1 Click File menu ► Publish.
- 2 In the Publish dialog box, click the Add Sheets button.
- 3 In the Select Drawings dialog box, select drawings. Click Select to add them to the list of sheets in the Publish dialog box.

NOTE You can also drag drawings from the desktop into the Publish dialog box to add sheets to the list.

All of the layouts in a drawing become individual sheets in the list of drawing sheets.

You must remove the drawing sheets that you don't want to become a part of the drawing set. Layouts must be initialized before they can be published. (A layout is initialized if its paper size is defined in the page setup to any size other than 0 x 0.)

Standard toolbar



Command line: PUBLISH

To add sheets from a DSD file to a drawing set for publishing

- 1 Click File menu ► Publish.
- 2 In the Publish dialog box, click the Load Sheet List button.
- 3 In the Load List of Sheets dialog box, select a DSD file. Click Load.
- 4 In the Replace or Append dialog box, click Replace to replace the current sheets with the sheets in the DSD file, or click Append to add the sheets in the DSD file to the list of sheets in the Publish dialog box.

Standard toolbar



Command line: PUBLISH

To remove a sheet from a drawing set for publishing

- 1 Click File menu ► Publish.
- 2 In the Publish dialog box, make sure the sheets that you want to remove are listed.
- 3 Under Sheets, select one or more sheets to remove. Click the Remove Sheets button.

Standard toolbar



Command line: PUBLISH

To remove all sheets from a drawing set for publishing

- 1 Click File menu ► Publish.
- 2 In the Publish dialog box, make sure the sheets that you want to remove are listed.
- 3 In the Sheet List area, right-click. Click Remove All.

NOTE Removing all sheets cannot be undone.

Standard toolbar



Command line: PUBLISH

To reorder sheets in a drawing set for publishing

- 1 Click File menu ► Publish.
- 2 In the Publish dialog box, make sure the sheets that you want to reorder are listed.

- 3 Select a sheet. Click either the Move Sheet Up or Move Sheet Down button.

NOTE Sheets in the drawing set are viewed or plotted in the order shown in the Publish dialog box.

Standard toolbar



Command line: PUBLISH

To copy sheets in a drawing set for publishing

- 1 Click File menu ► Publish.
- 2 In the Publish dialog box, make sure the sheets that you want to copy are listed.
- 3 Select one or more sheets in the Sheets list and right-click. Click Copy Selected Sheets.

The copied drawing sheets are added and highlighted at the end of the sheet list. When a sheet is copied, its name is created by the addition of *-copy(n)* at the end of the original sheet name. For example, if you create one copy of a sheet called *Plumbing*, the copied sheet is called *Plumbing-Copy(1)*. Each time you copy the same sheet, the *n* is incremented by 1. By creating copies of a sheet, you can have different page setups and other settings for the same sheet.

Standard toolbar



Command line: PUBLISH

To rename sheets in a drawing set for publishing

- 1 Click File menu ► Publish.
- 2 In the Publish dialog box, make sure the sheets that you want to rename are listed.
- 3 Select a sheet in the Sheet list, and then do one of the following:
 - Right-click. Click Rename Sheet.

■ Press F2.

- 4 Enter the new sheet name.

Standard toolbar



Command line: PUBLISH

To change the page setup of one sheet in a drawing set for publishing

- 1 Click File menu ► Publish.
- 2 In the Publish dialog box, make sure the sheet for which you want to change the page setup is listed.
- 3 Select a sheet in the Sheet list.
- 4 Under Page Setup, select a page setup to apply to the drawing sheet.

Standard toolbar



Command line: PUBLISH

To change the page setup of one or more sheets in a drawing set for publishing

- 1 Click File menu ► Publish.
- 2 In the Publish dialog box, make sure the sheets for which you want to change the page setups are listed.
- 3 Select one or more sheets in the list and right-click. Click Change Page Setup.
- 4 In the Page Setup list, select a page setup to apply to the drawing sheets.

NOTE It is important that you change the page setup for each layout depending on your desired output. Only model space page setups can be applied to model space sheets, and only paper space page setups can be applied to paper space sheets.

Standard toolbar



Command line: PUBLISH

To import a page setup from another drawing to apply to a drawing sheet for publishing

- 1 Click File menu ► Publish.
- 2 In the Publish dialog box, make sure the sheet for which you want to change the page setup is listed.
- 3 Select a sheet in the Sheet list.
- 4 Under Page Setup, select Import from the list.
- 5 In the Import Page Setups dialog box, select the drawing whose page setups you want to import. Click Import.

Standard toolbar



Command line: PUBLISH

To publish multiple layouts in a drawing

- 1 In the drawing area, click a layout tab that you want to publish.
- 2 Press and hold the CTRL key, and then click the other layout tabs that you want to publish.
- 3 Right-click one of the selected layout tabs. Click Publish Selected Layouts.
- 4 In the Publish dialog box, change any settings for the selected layouts.
- 5 Click Publish.

Command line: PUBLISH

Create a Paper or Plot File Drawing Set

You can publish a drawing set to a plotter or plot file.

You can assemble drawing sheets into a customized drawing set and publish the sheets to the plotter named in the page setup specified for each sheet. If

the plot device named in the page setup is a paper plotter, then your output will be a paper drawing set.

If the plotter is configured to plot to a file, the sheets are saved to files in the plot file location specified in the Publish Options dialog box. Each drawing sheet's plot file is saved with the same name as the sheet, with the appropriate file extension for the file (for example, *.plt*, *.jpg*, or *.bmp*). The default location can be changed in the Options dialog box, Plot and Publish tab, under Plot to File.

See also:

["Set Publish Options" on page 1050](#)

To create and publish a paper or plot file drawing set

- 1 With a drawing open, click File menu ► Publish.
- 2 In the Publish dialog box, the drawing layouts are shown in the Sheets List. To create the drawing set, modify the list of drawing sheets by doing any of the following as needed:

■ **Add sheets.** To add sheets from other drawings, click the Add Sheets button (or drag drawings from the desktop). In the Select Drawings dialog box, select drawings. Click Select to add them to the list of sheets in the Publish dialog box. All of the layouts in a drawing become individual sheets in the list of drawing sheets. You can remove the individual sheets if you do not want them to become part of the drawing set. A layout must be initialized (its paper size must be defined in the page setup to any size other than 0 x 0) before it can be published.

NOTE To include all layouts when you add sheets to a drawing set, make sure the Include Layouts When Adding Sheets option is selected on the shortcut menu or in the Publish dialog box.

■ **Include model layouts.** If an uninitialized (the paper size is not defined in the page setup or is set to 0 x 0) model layout is included, it will be marked as Uninitialized in the Status column on the sheet list. It can be plotted if a page setup override is applied.

NOTE To include the Model when you add sheets to a drawing set, make sure the Include Model When Adding Sheets option on the shortcut menu is selected.

- **Remove sheets.** To remove sheets from the list, select one or more sheets in the list. Click the Remove Sheets button. To remove all sheets, right-click. Click Remove All.
- **Reorder sheets.** To reorder the sheets up or down one position in the list, select a sheet. Click either the Move Sheet Up or Move Sheet Down button. Sheets in the drawing set are viewed or plotted in the order shown in the list.
- **Rename sheets.** To rename a sheet, select it in the list and right-click. Click Rename Sheet. Enter the new sheet name.
- **Change page setups.** To change the page setup for a layout, select the sheet. In the Page Setup list, select a named page setup from the list, or select one or more sheets in the list and right-click. Click Change Page Setup. In the Page Setup list, select a page setup or select Import to import page setups from another drawing or template. In the Import Page Setups dialog box, select a drawing with one or more page setups. Click Import. In the Page Setup list, select a named page setup.

NOTE It is important that you change the page setup for each layout depending on your desired output.

- **Copy sheets.** To copy one or more drawing sheets, select the sheets in the list and right-click. Click Copy Selected Sheets. The copied drawing sheets are added and highlighted at the end of the sheet list. When a sheet is copied, its name is created by the addition of *-Copy(n)* at the end of the original sheet name. For example, if you create one copy of a sheet called *Plumbing*, the copied sheet is called *Plumbing-Copy(1)*. Each time you copy the same sheet, the *n* is incremented by 1. By creating copies of a sheet, you can have different page setups and other settings for the same sheet.

- 3 When your list of drawing sheets is assembled and configured the way you want for your paper or plot file drawing set, click the Save Sheet List button.
- 4 In the Save List As dialog box, in the File Name box, enter a name for the drawing set list. Click Save.

The drawing set list is saved as a DSD (Drawing Set Descriptions) file.

- 5 In the Publish dialog box, under Publish To, select Plotter Named in Page Setup.
- 6 Click Publish to start the process.
If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress: paper drawings are being plotted or plot files are being created.
- 7 To view information about the processed publish job, right-click the plotter icon on the right side of the status bar. Click View Plot and Publish Details.

Standard toolbar



Command line: PUBLISH

Publish an Electronic Drawing Set

You can publish an electronic drawing set in the Design Web Format.

You can assemble drawing sheets into a customized electronic drawing set. An electronic drawing set is the digital equivalent of a set of plotted drawings. An electronic drawing set is saved as a single, multisheet DWF file and can be shared with customers, suppliers, or people within your own company who may need the drawings for examination or for their records.

You can also send the published electronic drawing set as an email attachment, share it using a project collaboration site such as Autodesk® Buzzsaw®, or post it to a website. Using Autodesk DWF Viewer, you can view or plot only the layouts you need.

You can also publish the drawing set as individual, single-sheet DWF files for each sheet.

When you publish to a DWF file, you use the *DWF6 ePlot.pc3* plotter configuration file. You can use the default *DWF6 ePlot.pc3* plotter driver as installed, or you can modify configuration settings such as color depth, display resolution, file compression, font handling, and other options. Once you modify the *DWF6 ePlot.pc3* file, all future plotting and publishing of DWF files will be affected.

WARNING Before you make changes to the original *DWF6 ePlot.pc3* file, make sure you have backed up a copy in case you want to return to the default settings.

See also:

["Set Publish Options" on page 1050](#)

To create an electronic drawing set

- 1 With a drawing open, click File menu ► Publish.

The Publish dialog box is displayed. If the Include Layouts When Adding Sheets option is selected either in the shortcut menu or in the Publish dialog box, all the layouts in the current drawing are listed in the sheet list.

- 2 In the Publish dialog box, you can modify the list of sheets by doing any of the following as needed:

■ **Add sheets.** To add sheets from other drawings, click the Add Sheets button (or drag drawings from the desktop). In the Select Drawings dialog box, select drawings. Click Select to add them to the list of sheets in the Publish dialog box. All of the layouts in a drawing become individual sheets in the list of drawing sheets. You must remove those sheets that you don't want to become a part of the drawing set. A layout must be initialized (its paper size must be defined in the page setup to any size other than 0 x 0) before it can be published.

NOTE To include all layouts when you add sheets to a drawing set, make sure the Include Layouts When Adding Sheets option is selected on the shortcut menu or in the Publish dialog box.

■ **Include model layouts.** If you include an uninitialized (the paper size is not defined in the page setup or is set to 0 x 0) model layout, it will be marked as Uninitialized in the Status column on the sheet list. It can be plotted if you select an override page setup for it in the Publish dialog box from the page setup drop-down list under Page Setups in the sheet list.

NOTE To include the model when you add sheets to a drawing set for a DWF file, make sure the Include Model When Adding Sheets option is selected either on the shortcut menu or in the Publish dialog box.

■ **Remove sheets.** To remove sheets from the list, select one or more sheets, and then click the Remove Sheets button. To remove all sheets, right-click. Click Remove All.

■ **Reorder sheets.** To reorder the sheets up or down one position in the list, select a sheet. Click either the Move Sheet Up or Move Sheet Down button. Sheets in the drawing set are viewed or plotted in the order shown in the list.

■ **Rename sheets.** To rename a sheet, select it in the list and right-click. Click Rename Sheet. Enter the new sheet name.

■ **Change page setups.** To change the page setup for a layout, select the sheet, and in the Page Setup list select a named page setup from the list; or select one or more sheets and right-click. Click Change Page Setup. In the Page Setup list, select a page setup, or select Import to import page setups from another drawing or template. In the Import Page Setups dialog box, select a drawing with one or more page setups. Click Import. In the Page Setup list, select a named page setup.

NOTE It is important that you change the page setup for each layout depending on your desired output. Only model space page setups can be applied to model space sheets, and only paper space page setups can be applied to paper space sheets.

■ **Copy sheets.** To copy one or more drawing sheets, select the sheets in the list and right-click. Click Copy Selected Sheets. The copied drawing sheets are added and highlighted at the end of the sheet list. When a sheet is copied, its name is created by the addition of *-Copy(n)* at the end of the original sheet name. For example, if you create one copy of a sheet called *Plumbing*, the copied sheet is called *Plumbing-Copy(1)*. Each time you copy the same sheet, the *n* is incremented by 1. By creating copies of a sheet, you can have different page setups and other settings for the same sheet.

- 3 When your list of drawing sheets is assembled and configured the way you want for your drawing set, click the Save List button.
- 4 In the Save List As dialog box, in the File Name box, enter a name for the list. Click Save.

The drawing set list is saved as a DSD (Drawing Set Descriptions) file.

- 5 In the Publish dialog box, under Publish To, select DWF File. Click Publish.
- 6 In the Select DWF File dialog box, enter a file name. Click Select to provide the name and destination for the DWF file.

NOTE You can also enter a URL so that the DWF file is uploaded to an FTP or HTTP site.

- 7 Click Save to start the electronic drawing set creation.
If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress.

- 8** To view information about the processed publish job, right-click the plotter icon on the right side of the status bar. Click View Plot and Publish Details.
The information in the Plot and Publish Details dialog box is also saved to the plot and Publish log file.
- 9** If you have background publishing enabled you can view the published DWF file. Right-click the plotter icon on the right side of the status. Click View DWF File.

Standard toolbar



Command line: PUBLISH

Publish a Sheet Set

From the Sheet Set Manager, you can easily publish an entire sheet set, a subset of a sheet set, or a single sheet. If you want to publish a sheet set that you've set up in the Sheet Set Manager, it's much quicker to publish the sheet set directly from the Sheet Set Manager, rather than starting from the Publish dialog box.

When you publish from the Sheet Set Manager, you can publish an electronic sheet set by publishing to a DWF file, or you can publish a paper set by publishing to the plotter named in the page setup that is associated with each drawing sheet. You can also publish your sheets using a page setup that is saved in the page setup overrides DWT file associated with the sheet set. This page setup overrides the current page setup settings for the individual publish job.

When you open the Publish dialog box from the Sheet Set Manager, the Publish dialog box automatically lists the sheets you selected in the sheet set. You can then modify the sheet set for publishing.

See also:

- ["Use Named Page Setups with Sheet Sets" on page 971](#)
- ["Work with Sheets in a Sheet Set" on page 239](#)
- ["Set Publish Options" on page 1050](#)

To publish a sheet set to a DWF file

- 1** In the Sheet Set Manager, under Sheets, select the Sheet Set node or name.

- 2** In the upper-right corner of the Sheet Set Manager, click the Publish to DWF button.



NOTE Sheet sets cannot contain 3D DWF entries.

If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress. Right-click this icon for options to view the DWF file or to view information about the publish job.

TIP Publishing performance can be improved by disabling background publishing. Background publishing can be disabled from the “Plot and Publish Tab (Options Dialog Box)” of the Options dialog box.

Standard toolbar



Command line: SHEETSET

Command line: Right-click a sheet set node or name. Click Publish ► Publish to DWF.

To plot a sheetset to a PDF file

- 1** In the Sheet Set Manager, right-click the sheet set or node name. Click Publish ► Manage Page Setups.
- 2** In the Page Setup Manager dialog box, click New.
- 3** In the New Page Setup dialog box, under New Page Setup Name, enter a name for the PDF page setup. Click OK.
- 4** In the Page Setup dialog box, under Printer/Plotter, select the *DWG to PDF.pc3* configuration file.
If the Paper Size Not Found dialog box displays, choose the paper size option for the sheet set.
- 5** In the Page Setup dialog box, set the desired page setup options. Click OK.
- 6** In the Page Setup Manager, click Close.
- 7** In the Sheet Set Manager, right-click the sheet set to publish. Click Publish ► Publish Using Page Setup Override ► page setup you created.

The PDF file is plotted to the directory listed in the Plot and Publish Details dialog box **VIEWPLOTDETAILS**.

TIP Publishing performance can be improved by disabling background publishing. Background publishing can be disabled from the “Plot and Publish Tab (Options Dialog Box)” of the Options dialog box.

Sheet Set Manager window



Command line: SHEETSET

To publish a subset of a sheet set or an individual sheet to a DWF file

- 1 In the Sheet Set Manager, under Sheets, select the node or name of a subset or an individual sheet.
- 2 In the upper-right corner of the Sheet Set Manager, click the Publish to DWF button.



NOTE If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress. Right-click this icon for options to view the DWF file or to view information about the publish job.

Standard toolbar



Command line: SHEETSET

Shortcut menu: Right-click the node or name of a subset or an individual sheet. Click Publish ➤ Publish to DWF.

To publish sheets in a sheet set to the plotter named in the sheet's page setup

- 1 In the Sheet Set Manager, under Sheets, select the node or name of a sheet set, a subset, or an individual sheet.
- 2 In the upper-right corner of the Sheet Set Manager, click the Publish button. Click Publish to Plotter.



NOTE If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress. Right-click this icon for options to view the DWF file or to view information about the publish job.

Standard toolbar



Command line: SHEETSET

Shortcut menu: Right-click the node or name of a sheet set, subset, or an individual sheet. Click Publish ► Publish to Plotter.

To publish sheets in a sheet set using a page setup from the page setup overrides file

- 1 In the Sheet Set Manager, under Sheets, select the node or name of a sheet set, subset, or an individual sheet.
- 2 In the upper-right corner of the Sheet Set Manager, click the Publish button. Click Publish Using Page Setup Override ► <filename.dwt>. (All the named page setups that are saved in the page setup override DWT file are listed.)



The settings in the selected page setup override file take precedence over (override) the page setup settings currently set for the selected sheets. The page setup override settings apply only for this individual publish job.

NOTE If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress. Right-click this icon for options to view the DWF file or to view information about the publish job.

Standard toolbar



Command line: SHEETSET

Shortcut menu: Right-click the node or name of a sheet set, subset, or an individual sheet. Click Publish ► Publish Using Page Setup Override ► <filename.dwt>.

To include a plot stamp on published sheets from a sheet set

- In the upper-right corner of the Sheet Set Manager, click the Publish button. Click Include Plot Stamp.



A plot stamp will be included on all published sheets.

NOTE To change the plot stamp settings, click the Publish button. Click Plot Stamp Settings.

Standard toolbar



Command line: SHEETSET

Shortcut menu: Right-click the node or name of a sheet set, subset, or an individual sheet. Click Publish ► Include Plot Stamp.

To set options for publishing sheet sets

- 1 In the upper-right corner of the Sheet Set Manager, click the Publish button. Click Sheet Set Publish Options.



- 2 In the Sheet Set Publish Options dialog box, modify the settings as required.
- 3 Click OK.

Standard toolbar



Command line: SHEETSET

Shortcut menu: Right-click the node or name of a sheet set, subset, or an individual sheet. Click Publish ➤ Sheet Set Publish Options.

To open the Publish dialog box from the Sheet Set Manager and modify the sheet set for publishing

- 1 In the Sheet Set Manager, under Sheets, select the node or name of a sheet set, subset, or an individual sheet.
- 2 In the upper-right corner of the Sheet Set Manager, click the Publish button. Click Publish Dialog Box.



The Publish dialog box is displayed and lists the sheets you selected in the Sheet Set Manager.

- 3 In the Publish dialog box, modify the sheet list and settings as required.
- 4 Click Publish to publish the modified sheet set.

NOTE If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress. Right-click this icon for options to view the DWF file or to view information about the publish job.

Standard toolbar



Command line: SHEETSET

Shortcut menu: Right-click the node or name of a sheet set, subset, or an individual sheet. Click Publish ➤ Publish Dialog Box.

Republish a Drawing Set

You can easily republish lists of drawing sheets that you previously saved in the Drawing Set Descriptions (DSD) file format.

After drawings have been updated, you may want to republish a drawing set for viewing or plotting. You can easily republish a sheet set (DST file) or a collection of drawing sheets that you've saved a description of as a DSD (Drawing Set Descriptions) file. You can also load and republish BP3 (Batch Plot) files.

To republish a drawing set

- 1 Click File menu ► Publish.

If you have a drawing open, all the layouts in your drawing are shown in the sheet list in the Publish dialog box. Right-click in the sheet list. Click Remove All to delete the layouts from the list of drawing sheets.

- 2 In the Publish dialog box, click the Load Sheet List button.
- 3 In the Load List of Sheets dialog box, select the location of the DSD file or the BP3 file. Click Load.
The drawing sheets in the saved drawing set are displayed in the sheet list in the Publish dialog box.
- 4 In the Publish dialog box, do one of the following:
 - If you want to publish the drawing set to a DWF file, under Publish To, select DWF file.
 - If you want to publish the drawing set to a plotter or printer, under Publish To, select Plotter Named in Page Setup.

- 5 Click Publish.

If you have background publishing enabled, the animated plotter icon on the right side of the status bar indicates that the publish job is in progress.

- 6 If you have background publishing enabled, you can view information about the processed publish job. Right-click the plotter icon on the right side of the status bar. Click View Plot and Publish Details. The information in the Plot and Publish Details dialog box is also saved to the plot and publish log file.

Standard toolbar



Command line: PUBLISH

View a Published Electronic Drawing Set

A published electronic drawing set can be viewed and plotted with Autodesk™ DWF™ Viewer.

A published drawing set is the digital equivalent of paper plots created from your original drawings. The drawing set, which is saved as a Design Web Format™ (DWF™) file, can be viewed or plotted by anyone using Autodesk DWF Viewer. With Autodesk DWF Viewer, you can open, view, and print all DWF file formats and other raster format images. You can pan, zoom, and view individual drawing sheets and viewports. You can also view layer information, sheet and sheet set properties, block information and attributes, and custom properties, if they are included in the DWF file. As you move your cursor across DWF geometry in the viewer, objects with associated data are displayed with a red highlighting effect.

Recipients of drawing sets in DWF format don't have to own or know the program. From anywhere in the world, they can view and print high-quality layouts using Autodesk DWF Viewer.

Autodesk DWF Viewer runs as a stand-alone application or embedded in any application that supports ActiveX controls, such as Microsoft® Internet Explorer.

For product information and a download link for the Autodesk DWF Viewer, refer to the Products page on the Autodesk website.

For more information about using Autodesk DWF Viewer, see Help in Autodesk DWF Viewer.

To view the most recently published DWF file in Autodesk DWF Viewer

- In the program, right-click the plotter icon on the right side of the status bar. Click View DWF file.

NOTE This option is available only if you have background publishing enabled.

Set Publish Options

You can set options for publishing, such as output file location, DWF type, multisheet DWF name options, DWF security (password protection), and whether or not to include layer information. You can also decide what types of information to reveal in your published DWF files.

You can include the following types of metadata:

- Sheet set properties (must publish using the Sheet Set Manager)
- Sheet properties (must publish using the Sheet Set Manager)
- Block standard properties and block custom properties and attributes
- Properties contained in custom objects

You use a block template (BLK) file to determine which blocks and properties to include in your published DWF file. You use the Block Template dialog box to create or modify the settings of a block template (BLK) file. You can also use BLK files created with the Attribute Extraction wizard.

When you change settings in the Publish Options dialog box, you can save the settings to the Drawing Set Descriptions (DSD) file to reuse the next time you publish drawings. You can also save them to just the DSD file.

To change the default output location for published DWF and plot files

- 1 Click File menu ► Publish.
- 2 In the Publish dialog box, under Publish To, click either Plotter Named in Page Setup or DWF File, and then click Publish Options.
- 3 In the Publish Options dialog box, under Default Output Location (DWF or Plot-to-File), do one of the following:
 - Click Location and make a selection from the drop-down list.
 - Click the [...] button, and in the Select a Folder for Generated Files dialog box, select a folder. Click Select.
- 4 In the Publish Options dialog box, click OK.

Standard toolbar



Command line: PUBLISH

To specify the DWF file output type to single-sheet DWF files or a single multi-sheet DWF file

- 1 Click File menu ► Publish.
- 2 In the Publish dialog box, under Publish To, select the DWF File, and then click Publish Options.

- 3 In the Publish Options dialog box, under General DWF Options, DWF Type, make a selection from the drop-down list as follows:

 - **Single-sheet DWF.** Specifies that an individual single-sheet DWF file is created for each sheet.
 - **Multi-sheet DWF.** Specifies that one multi-sheet DWF file is created.
- 4 Click OK.
- 5 In the Publish dialog box, either continue with publishing tasks or close the dialog box.

Standard toolbar



Command line: PUBLISH

To specify a name for multi-sheet DWF files

- 1 Click File menu ► Publish
- 2 In the Publish dialog box, under Publish To, click DWF File, and then click Publish Options.
- 3 In the Publish Options dialog box, under General DWF Options , DWF Type, select Multi-sheet DWF from the drop-down list.
- 4 In the Publish Options dialog box, under Multi-sheet DWF Options, DWF Naming, select Specify Name from the drop-down list.
- 5 Click OK.
- 6 In the Publish dialog box, either continue with publishing tasks or close the dialog box.

Standard toolbar



Command line: PUBLISH

To specify to be prompted for a name for a multi-sheet DWF file

- 1 Click File menu ► Publish

- 2** In the Publish dialog box, under Publish To, click DWF File, and then click Publish Options.
- 3** In the Publish Options dialog box, under General DWF Options, DWF Type, select Multi-sheet DWF from the drop-down list.
- 4** In the Publish Options dialog box, under Multi-sheet DWF Options, select Prompt for Name to be prompted for a file name every time you publish a DWF file.
- 5** Click OK.
- 6** In the Publish dialog box, either continue with publishing tasks or close the dialog box.

Standard toolbar



Command line: PUBLISH

To specify that DWF files are published with password security

- 1** Click File menu ► Publish.
- 2** In the Publish dialog box, under Publish To, click DWF File, and then click Publish Options.
- 3** In the Publish Options dialog box, under General DWF Options, Password Protection, select one of the following from the drop-down list:
 - Select Specify Password, and under Password, enter a password to use for the DWF file.
 - Select Prompt for Password. You will be prompted for a password or phrase to open the published DWF file.

DWF passwords are case-sensitive. The password or phrase can consist of letters, numbers, punctuation marks, or non-ASCII characters.

WARNING If you lose or forget the password, it cannot be recovered. Keep a list of passwords and their corresponding DWF file names in a safe place.

-
- 4** Click OK.
 - 5** In the Publish dialog box, either continue with publishing tasks or close the dialog box.

Standard toolbar



Command line: PUBLISH

To include layer information in a published DWF file

- 1 Click File menu ► Publish.
- 2 In the Publish dialog box, under Publish To, click DWF File, and then click Publish Options.
- 3 In the Publish Options dialog box, under DWF Data Options, Layer Information, click to display the drop-down list, and select Include.

NOTE By default, Layer Information is set to Don't Include, to reduce publishing time. If you change the setting to include layer information, you can turn individual layers on and off when you view or print the DWF file.

- 4 Click OK.
- 5 In the Publish dialog box, either continue with publishing tasks or close the dialog box.

Standard toolbar



Command line: PUBLISH

To include block information in a published DWF file

- 1 Click File menu ► Publish.
- 2 In the Publish dialog box, under Publish To, click DWF File, and then click Publish Options.
- 3 In the Publish Options dialog box, under DWF Data Options, Block Information, click to display the drop-down list, and select Include.

NOTE By default, Block Information is set to Don't Include. If you change the setting to include block information, you can use the viewer to view or print block property and attribute information in the DWF file.

- 4 Click OK.

- 5** In the Publish dialog box, either continue with publishing tasks or close the dialog box.

Standard toolbar



Command line: PUBLISH

To include block template information in a published DWF file

NOTE You can use block template (BLK) files created with the Publish Options dialog box or BLK files created using the Attribute Extraction wizard.

- 1** Click File menu ➤ Publish.
- 2** In the Publish dialog box, under Publish To, click DWF File. Click Publish Options.
- 3** In the Publish Options dialog box, under DWF Data Options, Block Information, click to display the drop-down list. Select Include.
- 4** In the Publish Options dialog box, under DWF Data Options, Block Template File, click to display the drop-down list. Select the block template (BLK) file that contains the properties and attribute definitions you want to make available in your published DWF file. Click OK.

NOTE The list also contains the Create and Edit options. Create opens the Publish Block Template dialog box where you can create a new block template file. Edit opens the Edit Publish Template dialog box where you can select an existing block template file to modify.

- 5** Click OK.
- 6** In the Publish dialog box, either continue with publishing tasks or close the dialog box.

Standard toolbar



Command line: PUBLISH

To create a block template (BLK) file

- 1 Click File menu ► Publish.
- 2 In the Publish dialog box, under Publish To, click DWF File, and then click Publish Options.
- 3 In the Publish Options dialog box, under DWF Data Options, Block Information, click to display the drop-down list, and select Include.
- 4 Under DWF Data Options, Block Template File, click to display the drop-down list, and select Create.

The Publish Block Template dialog box opens with the Block Source Drawing area displaying the current drawing.

NOTE The working set of drawings for this list is completely independent of the drawings to include in the Publish operation.

- 5 Do one of the following:
 - If this drawing contains the block properties you want to include in the published DWF file, click Scan for Blocks.
The program scans the DWG file for all unique block definitions and their associated properties and attributes.
 - If this drawing does not contain the block properties you want to include in the published DWF file, click Add.
The Select Drawings dialog box is displayed. Select drawings to add to the block source drawings list. You can also add and remove drawings that contain target blocks whose properties you want to include in the published DWF file. Click Scan for Blocks.
The DWG file is scanned for all unique block definitions and their associated properties and attributes.

NOTE By default, block settings for nested blocks and blocks in xrefs are included. If you don't want to include these block settings, click Options in the Publish Block Template dialog box and clear the related options.

- 6 In the Publish Block Template dialog box, under Block Data to Publish, Unique Blocks From Source Drawings, Check Blocks to Publish, select the block names you want to include in the published DWF files. You can right-click to select or clear all boxes.
- 7 Under Block Data to Publish, Properties of Selected Blocks, Check Properties to Publish, select the properties you would like to include in your published DWF files. You can right-click to select or clear all boxes.
The list displays the union of the properties of all selected blocks.

NOTE If you select a block on the block list and clear the check marks of all its properties, only the name of the block is published to the resultant DWF file; no property information is included.

- 8 Click Save to name and save the block template file. Click OK.
The new block template file is now available under DWF Data Options in the Publish Block Template dialog box.
- 9 In the Publish dialog box, either continue with publishing tasks or close the dialog box.

Standard toolbar



Command line: PUBLISH

To edit a block template (BLK) file

- 1 Click File menu ➤ Publish.
- 2 In the Publish dialog box, under Publish To, click DWF File, and then click Publish Options.
- 3 In the Publish Options dialog box, under DWF Data Options, Block Information, click to display the drop-down list, and select Include.
- 4 Under DWF Data Options, Block Template File, click to display the drop-down list, and select Edit.
- 5 In the Select Block Template dialog box, click the Block Template (BLK) file you want to edit, and then click Select.

NOTE The working set of drawings for this list is completely independent of the drawings to include in the Publish operation.

- 6 In the Publish Block Template dialog box, click Scan for Blocks.
The program scans the DWG files for all unique block definitions and their associated properties and attributes.

NOTE By default, block settings for nested blocks and blocks in xrefs are included. If you don't want to include these block settings, click Options in the Publish Block Template dialog box and clear the related options.

- 7** Under Block Data to Publish, Unique Blocks From Source Drawings, Check Blocks to Publish, select or clear the block names you want to include in the published DWF files.
- 8** Under Block Data to Publish, Properties of Selected Blocks, Check Properties to Publish, select or clear the properties you want to include in your published DWF files.

This list displays the union of the properties of all selected blocks.

NOTE If you select a block from the block list and clear the check marks of all its properties, only the name of the block is published to the resultant DWF file; no property information is included.

- 9** Click Save. Click OK.

The modified block template file is now available under DWF Data Options in the Publish Block Template dialog box.

- 10** In the Publish dialog box, either continue with publishing tasks or close the dialog box.

Standard toolbar



Command line: PUBLISH

3D DWF Publishing

You can create and publish DWF files of your three-dimensional models and view them with the Autodesk® DWF™ Viewer.

You can generate DWF files of your three-dimensional models with nearly the same visual fidelity as your original DWG files. You can create a single- or multi-sheet DWF file, which can contain both 2D and 3D model space objects. To access 3D DWF publishing, you use the 3DDWF, EXPORT or PUBLISH commands.

Recipients of 3D DWF files can view and print them using the Autodesk DWF Viewer. For product information and a download link for the Autodesk DWF Viewer, refer to the Products page on the Autodesk website.

For more information about using the Autodesk DWF Viewer, see the Autodesk DWF Viewer Help system.

Publish Single and Multiple 3D DWF Files

To publish a single 3D DWF file, you use the EXPORT or 3DDWF commands. With either of those commands running, the model you're currently working on is saved as a 3D DWF file.

To publish multiple 3D DWF files at one time, you use the PUBLISH command. The Publish dialog box displays a list of all the sheet names of the drawing files that are currently open. You can determine which sheets are published to a 3D DWF file by adding or removing sheets on the Sheet Name list.

NOTE The list of sheets displayed in the Sheet Name list of the Publish dialog box is controlled by the PUBLISHALLSHEETS system variable. By default, all sheets are displayed for all open drawings.

Publishing Materials

If you've assigned texture mapped materials to your models, those materials can be published with the 3D DWF file. Texture map orientation and scaling that is set in the drawing editor is retained in the published 3D DWF file.

There are some limitations to materials publishing.

- The Diffuse Map channel is the only mapping that gets published. If you use Opacity, Reflection, or Bump maps in your material, they do not get published.
- Procedural materials such as Wood or Marble do not get published.

NOTE Because the DWF Viewer and the rendering engine are different, you may encounter some deviation in how the texture maps appear in the DWF Viewer.

Section clipping and Xclipping were scoped but may return for Spago - BP
1/13/06

Section Clipping and XClipping

Your 3D model may contain a section object that may have active live sectioning. When a section object's live section functionality is enabled, the 3D model is sectioned.

If you publish a 3D DWF file when live sectioning is enabled, the resulting DWF file will show the clipped geometry. Geometry that is hidden by live sectioning will also be hidden in the DWF Viewer. Geometry that is shown with additional visual representation, such as dashed lines, is displayed in the same manner when viewed in the DWF Viewer. The section object is not visible in the viewer.

Xclipping is utilized in 3D DWF files. When an xref is clipped with XCLIP, the resulting 3D geometry in the 3D DWF file is also clipped according to the same profile. When a 3D object is clipped, the clipped side of the solid is shown with an additional face parallel to the xclip line. However, in the DWF Viewer, you cannot turn the visibility of clipped areas on or off.

Increase the Smoothness of 3D DWF Models

The smoothness of 3D DWF models is improved by changing the value of the 3DDWFPREC system variable. The setting for 3DDWFPREC ranges from 1 to 6. Higher settings greatly improve the appearance of objects in the DWF Viewer.

NOTE 3DDWFPREC is a global setting that affects all objects in the 3D model. Therefore, higher values have the potential of resulting in very large 3D DWF file sizes.

Improve Performance with Large Models

Every object in your model gets processed when a DWF file is published. When working with large models, you can greatly improve publishing performance by using blocks. For instance, your office model contains a basic cubicle arrangement of eight partition walls, a desk, and a chair. Those ten objects don't take much time to process by themselves. However, if that basic cubicle arrangement is used 100 times, it means that 1000 objects must be processed and the publishing time increases accordingly. If the basic cubicle arrangement is inserted into the model as a single block, those ten objects are processed once for each insertion.

You can improve publishing performance even more by grouping miscellaneous objects into temporary blocks. Once the publishing is complete, you can explode those blocks and continue working on those objects.

As a design gets closer to completion, you can create blocks comprised of portions of the model that are less likely to need further editing. If the first floor of the office model is not going to change much, create a block containing all of its components. During publishing, that block is processed as one object instead of each individual object on that floor.

3D DWF Unsupported Content

When publishing a 3D DWF file, some drawing content may not appear in the DWF Viewer. The content listed in the following table are not supported when you output a 3D DWF file.

Unsupported Content	Details
Animations and Walkthroughs	

Unsupported Content	Details
Font types (Various)	See the “Supported Text Fonts” table
Gradient fills (Hatches)	
Hidden edges	
Hyperlinks	
Images	
Layer information	
Lights and Shadows	
Material components	<ul style="list-style-type: none"> <li data-bbox="740 546 1190 587">■ Bump, Opacity, and Specular texture maps <li data-bbox="740 606 1190 648">■ Procedural materials (Wood & Marble) <li data-bbox="740 667 1190 708">■ Reflection and refraction <li data-bbox="740 727 1190 768">■ Self-Illumination <li data-bbox="740 787 1190 829">■ Shininess <li data-bbox="740 848 1190 889">■ Translucency
Mtext (partial)	Bold and Italic text is not supported
Named views and cameras	
OLE objects	
Rays and X-lines	
Section clipping and XClipping	
Text thickness	
Visual styles	

Supported Text Fonts

Many text fonts are not supported when you publish a 3D DWF file. The following table lists the fonts that will get published.

Supported Text Fonts

Arial

Supported Text Fonts

Arial Black

Comic Sans MS

Courier New

Impact

Lucinda Console

Lucinda Sans Unicode

Martlett

Tahoma

Times New Roman

Verdana

Verdana Italic

Webdings

Wingdings

To publish a single 3D DWF file

- 1 On the command line, enter **3ddwf**.
- 2 In the Export 3D DWF dialog box, specify a name and location of the DWF file. Click Save.
By default, all model space objects are published to the 3D DWF file and, if your drawing contains xrefs, the Group By Xref Hierarchy option is active.
- 3 (Optional) Click Yes to open the Autodesk DWF Viewer and view the published 3D DWF file.

Standard toolbar



Command line: 3DDWF

To export a 3D DWF file

- 1 Click File menu ► Export.

- 2 In the Export Data dialog box, specify the name and location of the DWF file.
- 3 If necessary, select 3D DWF (*.dwf) from the Files Of Type list. Click Save.

By default, all model space objects are exported to the 3D DWF file and if your drawing contains xrefs, the Group By Xref Hierarchy option is active.
- 4 (Optional) Click Yes to open the Autodesk DWF Viewer and view the published 3D DWF file.

Command line: EXPORT

To publish multiple 3D or combined 2D and 3D DWF files

- 1 Click File menu ➤ Publish.
- 2 In the Publish dialog box, under Sheets to Publish, select the sheets you want to publish and set their Page Setup / 3D DWF setting to 3D DWF.

The sheets that are displayed in the Sheets to Publish group are controlled by the PUBLISHALLSHEETS system variable.
- 3 In Publish To, select DWF file, if it's not already active.
- 4 Click Publish Options. In General DWF Options, set the DWF Type to Multi-sheet DWF.
- 5 Specify if you want materials to be published. Click OK.
- 6 Click Publish.

Standard toolbar



Command line: PUBLISH

To publish selected objects in your model to a 3D DWF file

- Click File ➤ Export.
- In the Export Data dialog box, specify a name and location for the DWF file.
- Click the Tools button at the upper right.
- Click Tools menu, ➤ Options.

- In the 3D DWF Publish dialog box, Objects To Publish, click Selected Model Space Objects and then click the Select Objects button.
- In the model, select the objects that you want to publish.
- Press ENTER or the SPACEBAR when you're done selecting objects.
- In the 3D DWF Publish dialog box, click OK.
- In the Export Data dialog box, click Save.
- (Optional) Click Yes to open the Autodesk DWF Viewer and view the published 3D DWF file.

Command line: EXPORT

To view and print a published 3D DWF file

- If you want to view a 3D DWF file, do one of the following:
 - Immediately after the 3D DWF file has been published, click Yes to the question "Would you like to view it now?" The Autodesk DWF Viewer opens the 3D DWF file.
 - In Windows Explorer, double-click the 3D DWF file to open the Autodesk DWF Viewer and view the file.
 - Open the Autodesk DWF Viewer. Click File menu ► Open. Select the 3D DWF file you want to view.
- If you want to print a 3D DWF file, in Autodesk DWF Viewer, click File menu ► Print. For more information, see the Autodesk DWF Viewer Help system.

Configure a DWF6 Driver (Advanced)

You are not likely to change the settings in the DWF6 ePlot Properties dialog box often. However, when you need to make some subtle changes to your DWF6 ePlot configuration, you can do so through the DWF6 ePlot Properties dialog box. You can specify settings for color depth, display resolution, file compression, handling of fonts, pen settings, and other options. When you create DWF6 files, you use a plotter configuration file that uses a DWF6 driver model. You must use the *DWF6 ePlot.pc3* file to plot DWF6 files.

Overview of Creating or Modifying a DWF6 Configuration File

Publish uses only the DWF6 *ePlot.pc3* plotter configuration file. You can use the default DWF6 *ePlot.pc3* plotter configuration file as installed, or you can make changes to it using the Properties button in the Plot dialog box. This invokes the pc3 editor, which allows you to make changes directly to the DWF6 *ePlot.pc3* file. Although you can save DWF6 *ePlot.pc3* files under other names for use with the Plot command, no other name will work with the Publish command. Any changes you make to the DWF6 *ePlot.pc3* file are used whenever you plot or publish DWF6 files until you change the settings of your DWF6 *ePlot.pc3* file again.

NOTE If you are going to make changes to the DWF6 *ePlot.pc3* file, make a backup copy of the file first in case you want to use the default settings later. If necessary, the default DWF6 *ePlot.pc3* file can be re-created with the Add-A-Plotter wizard.

These settings may change file size and plot quality, depending on the content of the source drawing for the Design Web Format™ (DWF™) file. You can specify the following settings when you edit the DWF6 *ePlot.pc3* configuration file:

- Color depth
- Display resolution
- Compression options
- Inclusion and handling of fonts
- Background color
- Virtual pen settings and patterns
- Inclusion of layer information
- Inclusion of a paper boundary
- Inclusion of a saved preview

NOTE DWF files intended for plotting should be configured with a white background color. If the background color is set to black, color 7 objects will be plotted as white. For all other background colors, color 7 objects will be plotted as black.

To create a plotter configuration file for DWF file output

- 1 Click File menu ► Plotter Manager.
- 2 Double-click the Add-A-Plotter wizard.
- 3 On the Add Plotter-Introduction page, click Next.
- 4 On the Add Plotter-Begin page, select My Computer. Click Next.
- 5 On the Plotter Model page, under Manufacturers, select Autodesk ePlot (DWF). Under Models, select the DWF6 ePlot file that you want to create. Click Next.
- 6 (Optional) If you want to import a pre-existing plotter configuration, on the Import PCP or PC2 page, click Import File. Select a PCP or PC2 file to import. Click Import.
- 7 Click Next.
- 8 On the Ports page, select Plot to File. Click Next.
- 9 On the Plotter Name page, enter a name for the plotter configuration file. Click Next.
- 10 On the Finish page, click Finish.

A new plotter configuration file (PC3) is created.

Command line: PLOTTERMANAGER

To specify or modify settings for plotted DWF files

- 1 Click File menu ► Plot.
- 2 In the Plot dialog box, under Printer/Plotter, in the Name list, select a DWF plotting device. Click Properties.
- 3 In the Plotter Configuration Editor, Device and Document Settings tab, select Custom Properties in the tree window.
- 4 Click Custom Properties.
- 5 In the DWF6 ePlot Properties dialog box, select the options you want. Click OK.
- 6 In the Plotter Configuration Editor, click OK.

- 7 In the Changes to a Printer Configuration dialog box, do one of the following. Then click OK.

 - Select Apply Changes for the Current Plot Only to specify a one-time override of the configuration settings that isn't saved to the ePlot configuration file.
 - Select Save Changes to the Following File to save the configuration changes to the DWF configuration file.
- 8 In the Browse for Plot File dialog box, in the Save In location list, select a plotting location for the DWF file, and click Save. Click OK.

Standard toolbar



Command line: PLOT

To set color depth for plotted DWF files

- 1 Click File menu > Plot.
- 2 In the Printer/Plotter area, in the Name list, select a DWF plotting device. Click Properties.
- 3 In the Plotter Configuration Editor, Device and Document Settings tab, expand the Graphics node in the tree window.
- 4 Click Vector Graphics.
- 5 In the Color Depth area, select a color depth. Click OK.
- 6 In the Changes to a Printer Configuration dialog box, do one of the following. Then click OK.

 - Select Apply Changes for the Current Plot Only to specify a one-time override of the configuration settings that isn't saved to the DWF configuration file.
 - Select Save Changes to the Following File to save the configuration changes to the DWF configuration file.
- 7 In the Browse for Plot File dialog box, in the Save In location list, select a plotting location for the DWF file, and click Save. Click OK.

Standard toolbar



Command line: PLOT

Set the DWF File Resolution

You can specify the pixel resolution for vector and raster graphics of the DWF6 files that you create. The higher the resolution, the greater the precision but the larger the file size.

WARNING Raster and gradient resolution settings can never exceed the vector setting.

The following list shows the default values for vector and raster resolutions:

- Vector resolution: 1200 dpi
- Custom vector resolution: 40000 dpi
- Gradient resolution: 200 dpi
- Custom gradient resolution: 200 dpi
- Color and grayscale resolution: 200 dpi
- Custom color and grayscale resolution: 200 dpi
- Black and white resolution: 400 dpi
- Custom black and white resolution: 400 dpi

When you create DWF files intended for plotting, select a resolution to match the output of your plotter or printer. High resolutions (over 2400 dpi) are for viewing. For example, when you create DWF files of drawings that contain a lot of detail, such as a topographical map of a large region, use a higher resolution setting for greater detail in the DWF file. Use extreme resolutions (over 40,000 dpi) only when necessary; they may produce very large files. As you increase the resolution setting, raster image quality increases, the speed of printing decreases, and memory requirements increase.

As an example of the difference between DWF resolution settings, consider a map of the world that you want to output as a DWF file for viewing. With a medium resolution setting, you might be able to zoom to a level of detail approximately the size of the state of California on the world map. With a high resolution setting, you might be able to zoom to a level of detail the size of a city. With an extreme resolution setting, you might be able to zoom to a level of detail the size of a building.

To specify the DWF file resolution

- 1 Click File menu ► Plot.
- 2 In the Printer/Plotter area, in the Name list, select a DWF plotting device. Click Properties.
- 3 In the Plotter Configuration Editor, Device and Document Settings tab, select Custom Properties from the tree window.
- 4 In the Access Custom Dialog area, click Custom Properties.

NOTE You can enter any whole number between 150 and 100,000,000 for color and grayscale resolution settings as long as the number does not exceed the current vector resolution setting.

- 5 In the DWF6 ePlot Properties dialog box, in the Vector and Gradient Resolution (Dots Per Inch) area, select Vector and Gradient Resolution settings from the list, or select Custom and enter custom settings. Click OK.
- 6 In the DWF6 ePlot Properties dialog box, in the Raster Image Resolution (Dots Per Inch) area, select Color and Grayscale Resolution settings or Black and White Resolution settings from the list, or enter Custom settings. Click OK.
- 7 In the Plotter Configuration Editor, click OK.
- 8 In the Changes to a Printer Configuration dialog box, do one of the following. Then click OK.
 - Select Apply Changes for the Current Plot Only to specify a one-time override of the configuration settings that isn't saved to the DWF configuration file.
 - Select Save Changes to the Following File to save the configuration changes to the DWF configuration file.
- 9 In the Browse for Plot File dialog box, in the Save In location list, select a plotting location for the DWF file, and click Save. Click OK.

Standard toolbar



Command line: PLOT

Set the DWF File Compression

By default, the DWF6 files that you create are in a compressed binary format. The compression does not result in any loss of data; this is the recommended output for most DWF files. You can also create zipped ASCII encoded 2D stream (plain text) files. You specify these settings when you create or edit a DWF configuration file.

To specify the DWF file compression

- 1 Click File menu ► Plot.
- 2 In the Printer/Plotter area, in the Name list, select a DWF plotting device. Click Properties.
- 3 In the Plotter Configuration Editor, Device and Document Settings tab, select Custom Properties from the tree window.
- 4 In the Access Custom Dialog area, click Custom Properties.
- 5 In the DWF6 ePlot Properties dialog box, in the Additional Output Settings and in the DWF Format area, specify a file compression option. Click OK.
- 6 In the Plotter Configuration Editor, click OK.
- 7 In the Changes to a Printer Configuration dialog box, do one of the following. Then click OK.
 - Select Apply Changes for the Current Plot Only to specify a one-time override of the configuration settings that isn't saved to the DWF6 configuration file.
 - Select Save Changes to the Following File to save the configuration changes to the DWF configuration file.
- 8 In the Browse for Plot File dialog box, in the Save In location list, select a plotting location for the DWF file, and click Save. Click OK.

Standard toolbar



Command line: PLOT

Set Font Handling for the DWF File

When you create DWF files, you can specify how fonts are handled and included in the DWF6 file. By default, font handling is set to Capture Some in the DWF6 ePlot Properties dialog box; you can specify which fonts are eligible for capture in your DWF file. This is the recommended option.

NOTE DWF file size can be affected by the font-handling settings as well as by the amount of text and the number and type of fonts used in the DWF file. If the size of your DWF file seems too large, try changing the font-handling settings.

To specify font handling for the DWF file

- 1 Click File menu > Plot.
- 2 In the Printer/Plotter area, in the Name list, select a DWF plotting device. Click Properties.
- 3 In the Plotter Configuration Editor, Device and Document Settings tab, select Custom Properties from the tree window.
- 4 In the Access Custom Dialog area, click Custom Properties.
- 5 In the DWF6 ePlot Properties dialog box, in the Font Handling area, select a font capture option. Click OK.
- 6 In the Plotter Configuration Editor, click OK.
- 7 In the Changes to a Printer Configuration dialog box, do one of the following. Then click OK.
 - Select Apply Changes for the Current Plot Only to specify a one-time override of the configuration settings that isn't saved to the DWF6 configuration file.
 - Select Save Changes to the Following File to save the configuration changes to the DWF configuration file.
- 8 In the Browse for Plot File dialog box, in the Save In location list, select a plotting location for the DWF file, and click Save. Click OK.

Standard toolbar



Command line: PLOT

To edit the list of fonts eligible for capture in the DWF file

- 1 Click File menu ► Plot.
- 2 In the Printer/Plotter area, in the Name list, select a DWF plotting device. Click Properties.
- 3 In the Plotter Configuration Editor, Device and Document Settings tab, select Custom Properties from the tree window.
- 4 In the Access Custom Dialog area, click Custom Properties.
- 5 In the DWF6 ePlot Properties dialog box, in the Font Handling area, select Capture Some.
- 6 Select Edit Font List.

NOTE Capturing fonts in the DWF file causes the file size to increase. To minimize file size, the TrueType fonts that are common to all Microsoft® Windows® platforms are by default not selected in the list. Even though these fonts have no check marks beside them, they are shown in the DWF6 file if you are using a Windows operating system. You should select all other TrueType fonts that you have installed to ensure that they are captured for embedding in the DWF6 file. Only the fonts that are needed are embedded in the file.

- 7 In the Available TrueType Fonts dialog box, select the fonts that you want to be eligible for capture in the DWF file. Click OK.
- 8 In the DWF6 ePlot Properties dialog box, click OK.
- 9 In the Plotter Configuration Editor dialog box, click OK.
- 10 In the Browse for Plot File dialog box, in the Save In location list, select a plotting location for the DWF file, and click Save. Click OK.

Standard toolbar



Command line: PLOT

Edit Pen Patterns for the DWF File

In the Edit Pen Set dialog box, you can specify pen pattern, width, shape, effect, and other settings such as scaling and global pen widths. The changes you make in the Edit Pen Set dialog box are saved in the plotter configuration file.

NOTE To edit pen patterns when using DWF6 ePlot, you must select 255 Virtual Pens as your color depth.

To edit the Edit Pen Set dialog box for the DWF file

- 1 Click File menu ➤ Plot.
- 2 In the Printer/Plotter area, in the Name list, select a DWF plotting device. Click Properties.
- 3 In the Plotter Configuration Editor, Device and Document Settings tab, expand the Graphics node in the tree window. Select Vector Graphics. In the Color Depth Area, select 255 Virtual Pens as your color depth, and then select Custom Properties from the tree window.
- 4 In the Access Custom Dialog area, click Custom Properties.
- 5 In the DWF6 ePlot Properties dialog box, click Edit Patterns.
- 6 In the Edit Pen Set dialog box, right-click a field.
- 7 Select from the menu of common settings, or click Properties to display the Pen Properties dialog box where you can modify the attributes for each individual pen set.
- 8 In the Pen Properties dialog box, click OK.
- 9 In the Edit Pen dialog box, click OK.
- 10 In the DWF6 ePlot Properties dialog box, click OK.
- 11 In the Plotter Configuration Editor dialog box, click OK.
- 12 In the Changes to a Printer Configuration File dialog box, click one of the following:
 - Apply Changes for the Current Plot Only.
 - Save Changes to the Following File. (Enter the file name.)
- 13 In the Browse for Plot File dialog box, in the Save In location list, select a plotting location for the DWF file, and click Save. Click OK.

Standard toolbar



Command line: PLOT

Part 9

Share Data Between Drawings and Applications

Chapter 32 Reference Other Drawing Files

32

Reference Other Drawing Files

You can attach an entire drawing to the current drawing as a referenced drawing (*xref*). With xrefs, changes in the referenced drawing are reflected in the current drawing. Attached xrefs are linked to, but not actually inserted in, another drawing. Therefore, with xrefs you can build drawings without significantly increasing the drawing file size.

In this chapter

- Overview of Referenced Drawings (Xrefs)
- Attach and Detach Referenced Drawings
- Update and Bind Referenced Drawings
- Understand Clipping Boundaries
- Edit Referenced Drawings
- Resolve Referenced Drawing Errors
- Increase Performance with Large Referenced Drawings

Overview of Referenced Drawings (Xrefs)

By using referenced drawings, you can

- Coordinate your work with the work of others by referencing other drawings in your drawing to keep up with the changes being made by other designers. You can also assemble a master drawing from component drawings that may undergo changes as a project develops.
- Ensure that the most recent version of the referenced drawing is displayed. When you open your drawing, each referenced drawing is automatically reloaded, so it reflects the latest state of the referenced drawing file.
- Keep the names of layers, dimensioning styles, text styles, and other named elements in your drawing separate from those in referenced drawings.
- Merge (bind) attached referenced drawings permanently with your current drawing when the project is complete and ready to be archived.

NOTE Like a block reference, an xref appears in the current drawing as a single object. However, you cannot explode an xref without binding it first.

See also:

["DesignCenter" on page 43](#)

Attach and Detach Referenced Drawings

You can perform several operations on referenced drawing (xref) files. Start with attaching, setting paths, and detaching.

Attach Drawing References (Xrefs)

When you attach a drawing as an *xref*, you link that referenced drawing to the current drawing; any changes to the referenced drawing are displayed in the current drawing when it is opened or reloaded.

A drawing 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.

NOTE The objects in the xref attached to the current drawing are model space objects only. You can insert the xref into the current drawing in either model space or paper space. The xref can be attached at any location, scale, and rotation.

Tools for Attaching Xrefs

To attach an xref, use the EXTERNALREFERENCES command to display the External References palette, select DWG Reference from the View menu, or use XATTACH.

You can also use DesignCenter™ to attach xrefs to a drawing. Use DesignCenter for simple attachments, previewing drawing references and their descriptions, and quick placement by dragging.

You can attach an xref by dragging it from DesignCenter or by clicking Attach as Xref on the shortcut menu.

The saved path used to locate the xref can be an absolute (fully specified) path, a relative (partially specified) path, or no path.

If an xref contains any variable block attributes, they are ignored.

NOTE When using the External References palette, it is recommended that you turn on the Auto-hide feature or anchor the palette. The palette will then hide automatically when you specify the insertion point of the external reference.

Notification of Attached Xrefs

An xref icon is displayed in the lower-right corner of the application window (the status bar tray) when xrefs are attached to the drawing.



When one or more xrefs are not found or if any of the xrefs need reloading, an exclamation point is added to the Xref icon. If you click the Xref icon, the External References palette is displayed.

Visibility, Color, and Linetype

You can control the visibility, color, linetype, and other properties of an xref's layers and make these changes temporary or permanent. If the 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.

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.

Operation in Single Document Interface Mode

Managing xrefs is handled differently when you run in a single document interface mode.

If you choose to operate in single document interface (SDI) mode, the External References palette is not available for managing file references that are attached to the drawing. Single document interface mode is set with the SDI system variable.

If the External References palette is visible when SDI mode is enabled, the External References palette remains visible but becomes inactive. The only control you have is to close or hide the palette.

Attempting to open the External References palette results in opening the legacy Xref Manager dialog box. The XREF command will also open the Xref Manager dialog box. The External References palette can be reinstated by setting the SDI system variable to 0 (zero).

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 1088
- “Clip Referenced Drawings and Blocks” on page 1093
- “Set Paths to Referenced Drawings” on page 1082
- “DesignCenter” on page 43
- “Set Interface Options” on page 62

To attach an xref

- 1 Click Insert menu ▶ DWG Reference.
- 2 In the Select Reference File dialog box, select the file you want to attach and then click Open.
- 3 In the External Reference dialog box, under Reference Type, select Attachment.
- 4 Specify the insertion point, scale, and rotation angle. Click Specify On-Screen to use the pointing device.

Attachment includes all nested xrefs.

- 5 Click OK.

Reference toolbar



Command line: XATTACH

To attach or overlay an xref using DesignCenter

- 1 Click Tools menu ► Palettes ► DesignCenter.
- 2 In the content area or in the Search dialog box, locate the DWG reference you want to attach or overlay.
- 3 Right-click. Drag the DWG reference into the open drawing.
- 4 Release the right-pointing device button. Click Attach as Xref.
- 5 In the External Reference dialog box, under Reference Type, select Attachment or Overlay.
- 6 Enter values for Insertion Point, Scale, and Rotation, or select Specify On-Screen to use the pointing device.
- 7 Click OK.

You can also attach an xref by dragging or by clicking Attach as Xref on the shortcut menu.

- 8 Click OK.

Standard toolbar



Command line: ADCENTER

To view referenced files by type in the External References palette

- 1 Click Insert menu ► External References.
- 2 In the External References palette, in the File References pane, click the List View button.
- 3 Click the Icon column.

The attached referenced files are sorted by type in the list view.

Reference toolbar



Command line: EXTERNALREFERENCES

Set Paths to Referenced Drawings

You can view and edit the file name and path that the program uses when locating a drawing reference. 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: an absolute path, a relative path, and no path.

Specify an Absolute Path

An absolute path is a fully specified hierarchy of folders that locates the file reference. An absolute path includes a local hard drive letter, a URL to a website, or a network server drive letter. This is the most specific but least flexible option.

Specify a Relative Path

Relative paths are partially specified folder paths that assume the current drive letter or the folder of the host drawing. This is the most flexible option, and enables you to move a set of drawings from your current drive to a different drive that uses the same folder structure.

If the file that is being referenced is located on a different local hard drive or on a network server, the relative path option is not available.

The conventions for specifying a relative folder path are as follows:

\

Look in the root folder of the host drawing's drive

path

From the folder of the host drawing, follow the specified path

path

From the root folder, follow the specified path

.*path*

From the folder of the host drawing, follow the specified path

..\path

From the folder of the host drawing, move up one folder level and follow the specified path

..\..\path

From the folder of the host drawing, move up two folder levels and follow the specified path

NOTE If a drawing that contains referenced files is moved or saved to a different path, to a different local hard drive, or to a different network server, you must edit any relative paths to accommodate the host drawing's new location or you must relocate the referenced files.

Specify No Path

When no path information is saved with the attached external reference, the following search is initiated in the order shown:

- Current folder of the host drawing
- Project search paths defined on the Files tab in the Options dialog box and in the PROJECTNAME system variable
- Support search paths defined on the Files tab in the Options dialog box
- Start In folder specified in the Microsoft® Windows® application shortcut

Specifying the No Path option is useful when moving a set of drawings to a different folder hierarchy or to an unknown folder hierarchy.

Know when a Referenced Drawing has been Relocated

If the drawing you are working on contains an xref that has been moved to a different folder, a message is displayed at the site of the xref when you load the drawing. The message indicates that the xref cannot be loaded using the old path. When you specify the new path, the xref is reloaded into your drawing.

Use Project Names to Define Referenced Drawing Paths

Project names make it easier for you to manage xrefs when drawings are exchanged between customers or if you have different drive mappings to the same location on a server. The project name points to a section in the registry that can contain one or more search paths for each project name defined.

If the program cannot find an xref at the location specified by the search path, the prefix (if any) is stripped from the path. If the drawing has a

PROJECTNAME value set and a corresponding entry exists in the registry, the program searches for the file along the project search paths. If the xref still is not located, the program search path is searched again.

You can add, remove, or modify the project names that exist in the registry. The folder search paths beneath the project name can also be added, removed, or modified.

The search paths beneath the project name can be added, removed, or modified in the same manner as the project name. The order in which the folders are searched can also be modified. Projects and their search paths can only be edited through the Files tab in the Options dialog box. You cannot edit project names on the command line.

Once you have established a project name and the search paths you want associated with that project name, you can make that project name the currently active project. The program searches the paths associated with that currently active project for xrefs that were not found in the full search path, the current drawing folder, or the program support paths.

Change Reference Paths in Multiple Drawings (Advanced)

When you relocate drawing files or the external files that they reference, including other drawing files, text fonts, images, and plot configurations, the reference paths that are saved in the drawing need to be updated. The Autodesk Reference Manager lists referenced files in selected drawings and provides tools to modify the saved reference paths without having to open each drawing file. You can also use Reference Manager to identify and fix unresolved references.

Reference Manager is a stand-alone application with its own Help system. Access is from the Start menu (Windows) ► (All) Programs ► Autodesk ► AutoCAD ► Reference Manager. If you are working on a network, it may not have been installed on your computer.

WARNING Reference Manager is intended primarily for CAD managers who thoroughly understand paths. It changes multiple paths at once, and you cannot undo your actions. Use extreme caution, especially if you are working on a network.

To change the path of a DWG reference

- 1 Click Insert menu ► External References.
- 2 In the External References palette, select a DWG reference name.
- 3 Under Found At, do one of the following:
 - Edit the xref path directly.

■ Click Browse. Select the xref in its new path.

4 Click OK.

The program reloads the xref and then regenerates the drawing with the xref in place.

Reference toolbar



Command line: EXTERNALREFERENCES

To display the currently defined project names

1 Click Tools menu ► Options.

2 In the Options dialog box, Files tab, double-click Project Files Search Path.

3 Click each project name folder to display the search paths associated with it.

4 Click OK (or Apply).

To add a project name

1 Click Tools menu ► Options.

2 In the Options dialog box, Files tab, double-click Project Files Search Path. Click Add.

A folder named *projectx* (where *x* indicates the next available number) is created and indented beneath the project folder.

3 Either enter a new name, or press ENTER to accept *projectx*.

The project name must be 31 characters or fewer, and it cannot contain leading spaces or terminating spaces.

4 Click OK (or Apply).

To remove a project name

1 Click Tools menu ► Options.

2 In the Options dialog box, Files tab, double-click Project Files Search Path.

3 Select a project name. Click Remove.

- 4 Click OK (or Apply).

To modify a project name

- 1 Click Tools menu ► Options.
- 2 In the Options dialog box, Files tab, select a project name. Enter a new name.
- 3 Click OK (or Apply).

You can also modify a project name by selecting the name in the project folder and pressing F2.

To add a search path

- 1 Click Tools menu ► Options.
- 2 In the Options dialog box, Files tab, select a project name. Click Add.
- 3 Add a new search path beneath the project name by entering a new path, or click Browse and select a new path.
- 4 Click OK (or Apply).

The new path is indented and placed beneath the project name.

To delete a search path

- 1 Click Tools menu ► Options.
- 2 In the Options dialog box, Files tab, select a project name. Click Remove.
- 3 Click OK.

To change a search path

- 1 Click Tools menu ► Options.
- 2 In the Options dialog box, Files tab, select a project name. Click Browse.
- 3 In the Browse for Folder dialog box, select a new path.
- 4 Click OK to close each dialog box.

You can also change a search path by selecting the project path and pressing F2.

To make a project current

- 1 Click Tools menu ▶ Options.
- 2 In the Options dialog box, Files tab, double-click Project Files Search Path.
- 3 Select a Project name. Click Set Current.
- 4 Click OK (or Apply).

You can also set a project current by entering **projectname** on the command line and then entering the name of the project.

To clear the current project

- 1 Click Tools menu ▶ Options.
 - 2 In the Options dialog box, Files tab, click Project Files Search Path.
 - 3 Click Clear Current.
This clears the setting for the current drawing.
 - 4 Click OK (or Apply).
- You can also clear the current project at the Command prompt by entering **projectname** and then entering a period (.).

Detach Referenced Drawings

To completely remove xrefs from your drawing, you need to detach them. Erasing xrefs does not remove, for example, layer definitions associated with those xrefs. Using the Detach option removes the xrefs and all associated information.

To detach an xref

- 1 Click Insert menu ▶ External References.
- 2 In the External Reference palette, select a DWG reference.
- 3 Right-click the selected DWG reference and select Detach from the shortcut menu.

Reference toolbar



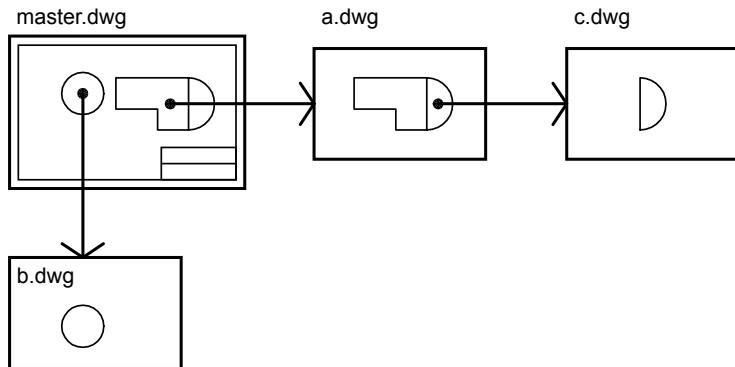
Update and Bind Referenced Drawings

Beyond attaching and detaching referenced drawings (xrefs), you can also update and bind drawing references.

Nest and Overlay Referenced Drawings

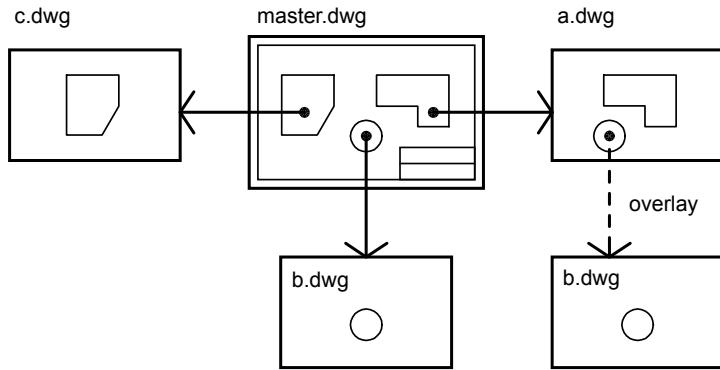
Xrefs can be nested within other xrefs: that is, you can attach an xref that contains another xref. You can attach as many copies of an xref as you want, and each copy can have a different position, scale, and rotation.

In the following illustration, *master.dwg* references *a.dwg* and *b.dwg*. Drawing *a.dwg* references *c.dwg*. In *master.dwg*, *c.dwg* is a nested xref.



You can also overlay an xref on your drawing. Unlike an attached xref, an overlaid xref is *not* included when the drawing is itself attached or overlaid as an xref to another drawing. Overlaid xrefs are designed for data sharing in a network environment. By overlaying an xref, you can see how your drawing relates to the drawings of other groups without changing your drawing by attaching an xref.

In the following illustration, several people are working on drawings referenced by *master.dwg*. The person working on *a.dwg* needs to see the work being completed by the person working on *b.dwg*, but does not want to xref *b.dwg* because it would then appear twice in *master.dwg*. Instead, the person overlays *b.dwg*, which is not included when *a.dwg* is referenced by *master.dwg*.



Relative Saved Paths and Nested Xrefs

The saved path for an xref can be an absolute (fully specified) path, a relative (partially specified) path, or no path. For a nested xref, a relative path always references the location of its immediate host and not necessarily the currently open drawing.

To overlay an xref

- 1 Click Insert menu > DWG Reference.
- 2 In the Select Reference File dialog box, select the file you want to attach and then click Open.
- 3 In the External Reference dialog box, under Reference Type, select Overlay.
- 4 Specify the insertion point, scale, and rotation angle. Click Specify On-Screen to use the pointing device.
Using the Overlay option excludes any nested xrefs.
- 5 Click OK.

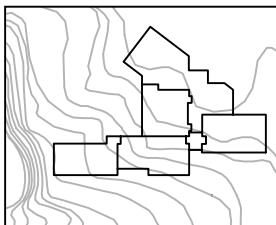
Reference toolbar



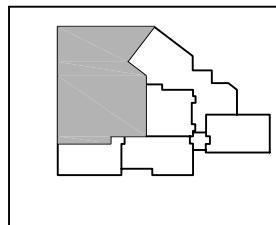
Command line: XATTACH

Update Referenced Drawing Attachments

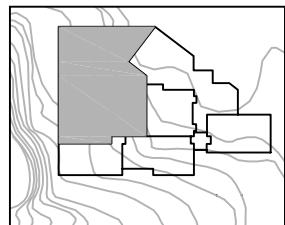
When you open a drawing, all xrefs update automatically. Use the Reload option from the External References palette to update xrefs whenever you want to ensure that the most current versions are displayed in your drawing.



xref attached



edited xref file



xref updated

Whenever you modify and save an externally referenced drawing in a network environment, other people can access your changes immediately by reloading the xrefs in their open drawings.

Notification of Changed Xrefs

When you attach xrefs to a drawing, the program periodically checks whether the referenced files have changed since the last time the xrefs were loaded or reloaded. The XREFNOTIFY system variable controls xref notification.

By default, if a referenced file has changed, a balloon message is displayed near the Xref icon in the lower-right corner of the application window (the status bar tray). Click the link in the balloon to reload all changed xrefs.



If you close the balloon message without reloading, an exclamation point is added to the Xref icon. If you click the Xref icon, the External References palette is displayed.

By default, the program checks for changed xrefs every five minutes. You can change the number of minutes between checks by setting the XNOTIFYTIME

system registry variable using (**setenv "XNOTIFYTIME" " n "**) where *n* is a number of minutes between 1 and 10080 (seven days).

NOTE When changing the value of XNOTIFYTIME, you must enter **XNOTIFYTIME** with the capitalization as shown.

Update Xrefs with Demand Loading Turned On

If demand loading is turned on when you load or reload an xref

- With the XLOADCTL system variable set to 1, the referenced drawing is kept open and locked. No one else can modify the referenced drawing.
- With XLOADCTL set to 2, a temporary copy of the most recently saved version of the referenced file is opened and locked. Others can open and modify the referenced drawing.

For information about demand loading, see “Increase Performance with Large Referenced Drawings” on page 1109.

To update an attached xref

- 1 Click Insert menu ▶ External References.
- 2 In the External References palette, select the reference name that you want to reload.
- 3 Right-click, and click Reload.

NOTE If the drawing you selected has been changed since you opened your drawing, the xref is reloaded.

Reference toolbar



Command line: EXTERNALREFERENCES

Archive Drawings That Contain Referenced Drawings (Bind)

When you archive final drawings that contain xrefs, you have two choices:

- Store the xref drawings along with the final drawing

- Bind the xref drawings to the final drawing

Storing an xref drawing along with the final drawing requires that the drawings always remain together. Any change to the referenced drawing will continue to be reflected in the final drawing.

To prevent unintentional updating of archived drawings by later changes to referenced drawings, bind the xrefs to the final drawing.

Binding an xref to a drawing makes the xref a permanent part of the drawing and no longer an externally referenced file. You can bind the entire database of the xref drawing, including all its xref-dependent named objects (blocks, dimension styles, layers, linetypes, and text styles), by using the XREF Bind option. For more information, see “Resolve Name Conflicts in External References” on page 1105.

Binding xrefs to a drawing is also an easy way to send a drawing to reviewers. Rather than sending a master drawing plus each of the drawings it references, you can use the Bind option to merge the xrefs into the master drawing.

NOTE You cannot bind xrefs that contain proxy objects. For more information, see “Work with Custom and Proxy Objects”.

To bind an xref to the current drawing

- 1 Click Insert menu ► External References.
- 2 In the External References palette, select the reference name that you want to bind.
- 3 Right-click, and click Bind.
- 4 In the Bind Xrefs dialog box, select one of the following options:
 - Bind converts the objects in the xref into a block reference. Named object definitions are added to the current drawing with a prefix of blockname\$n\$.
 - Insert also converts the objects in the xref into a block reference. Named object definitions are merged into the current drawing without adding prefixes.
- 5 Click OK to close each dialog box.

Reference toolbar



Understand Clipping Boundaries

Use clipping boundaries to display specific parts of a referenced drawing attachment or block insertion.

Clip Referenced Drawings and Blocks

After attaching a drawing as an xref or inserting a block, you can define a clipping boundary by using XCLIP. A clipping boundary can define a portion of a block or xref while suppressing the display of geometry outside of the boundary. Clipping applies to an individual instance of an xref, not to the xref definition itself. The portion of the xref or block within the clipped boundary remains visible, and the remainder of the xref or block becomes invisible. The referenced geometry is not altered; only the display of the xref is edited.

You can use XCLIP to create a new clipping boundary, delete an existing boundary, or generate a polyline object coincident with the vertices of the clipping boundary. Xref clipping can be turned on or off. When a clipping boundary is turned off, the boundary is not displayed and the entire xref is visible, provided that the geometry is on a layer that is on and thawed. When a clipping boundary is turned off, it still exists and can be turned on. However, deleting a clipping boundary is permanent.

After an xref or block has been clipped, it can be edited, moved, or copied just like an unclipped xref or block. The boundary moves with the reference. If an xref contains nested clipped xrefs, they appear clipped in the drawing. If the parent xref is clipped, the nested xrefs are also clipped.

If you want to see the clipping boundary, you can turn on the XCLIPFRAME system variable. XCLIPFRAME determines whether the clipping boundary frame is displayed. When the clipping frame is on (set to 1), it can be selected as part of the object and plotted.

Clipping Boundary Options

An xref clipping boundary can be specified as a rectangular window or a polygonal boundary. You can also select a polyline to define the clipping boundary. The boundary can be specified anywhere in 3D space, but it is always applied planar to the current UCS. If a polyline is selected, the clipping boundary is applied in the plane of that polyline.

No procedures for this topic.

Define Clipping Boundaries

Rectangular Window Clipping Boundary

When you specify a rectangular clipping boundary, you are prompted for the corners of the window. The window is drawn parallel to the current UCS, and the clipping boundary is applied normal to the plane it lies on.

Polygonal Clipping Boundary

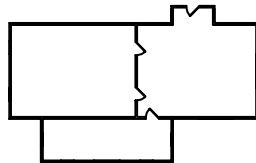
When you specify a polygonal clipping boundary, you are prompted to enter points to define the boundary. As you specify the clipping points, the last segment of the polygon is sketched so that the boundary is closed at all times. When you apply polygonal clipping to images in externally referenced drawings, the clipping boundary is applied to the rectangular extents of the polygonal boundary, rather than to the polygon itself.

Polyline Clipping Boundary

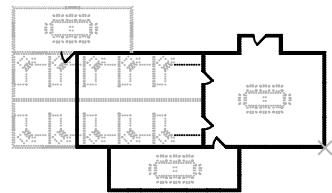
When you specify a polyline clipping boundary, you are prompted to select a 2D polyline object. The clipping boundary is created coincident with the polyline. Valid boundaries are 2D polylines with straight or spline-curved segments. Polylines with arc segments, or fit-curved polylines, can be used as the definition of the clip boundary, but the clip boundary will be created as a straight segment representation of that polyline. If the polyline has arcs, the clip boundary is created as if it had been decurved prior to being used as a clip boundary. An open polyline is treated as if it were closed.

Clip Depth

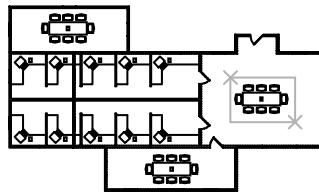
The front and back clipping planes for xrefs can be set. The xref must contain a clipping boundary before you can specify a *clip depth*. The clip depth is always calculated normal to the clipping boundary. When defining clip depth, you are prompted to specify a front and back point or distance relative to the clipped plane. Regardless of the current UCS, the clip depth is applied parallel to the clipping boundary.



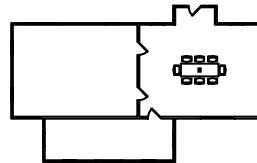
existing drawings



another similar drawing is referenced



xref is clipped to show only the conference table



resulting clipped xref

To clip a reference

- 1 On the command line, enter **xclip**.



- 2 Select a reference.
- 3 At the prompt, select New Boundary by pressing ENTER.
- 4 Select a rectangular or polygonal clipping boundary. Specify the corners or vertices of the boundary.

The image is clipped based on the area that you specified and hides the portion of the xref outside the clipping boundary.

Command line: XCLIP

Edit Referenced Drawings

Referenced drawings can be edited by opening them directly, or you can edit the xref in place from within the current drawing. You can edit a block definition directly from any selected block reference.

Edit a Referenced Drawing in a Separate Window

The simplest and most direct method for editing xrefs is to open the referenced drawing file in a separate window. This method gives you access to all objects in the referenced drawing.

Instead of browsing for the xref using the Select File dialog box, you can select the xref and open the referenced drawing. To use this method to edit xrefs, use the External References palette or the XOPEN command.

To edit an xref in a separate window

- 1 Click Insert menu ► External References.
- 2 In the External References palette, select the reference name that you want to edit.
- 3 Right-click, and click Open.
- 4 The selected drawing reference is opened in a new window where you can edit the drawing, save it, and then close the drawing.

Reference toolbar



Command line: EXTERNALREFERENCES

Edit Selected Objects in Referenced Drawings and Blocks

You can modify external references and redefine block definitions from within the current drawing by using in-place reference editing. Both blocks and xrefs are considered references.

By editing the reference in place, you can modify the reference within the visual context of your current drawing.

Often, a drawing contains one or more xrefs as well as multiple block references. When working with block references, you can select a block, modify it, view and edit its properties, and update the block definition. You cannot edit a block reference that is inserted using MINSERT.

When working with xrefs, you can select the reference you want to work with, modify its objects, and save back the changes to the reference drawing. You can make minor changes without having to go back and forth between drawings.

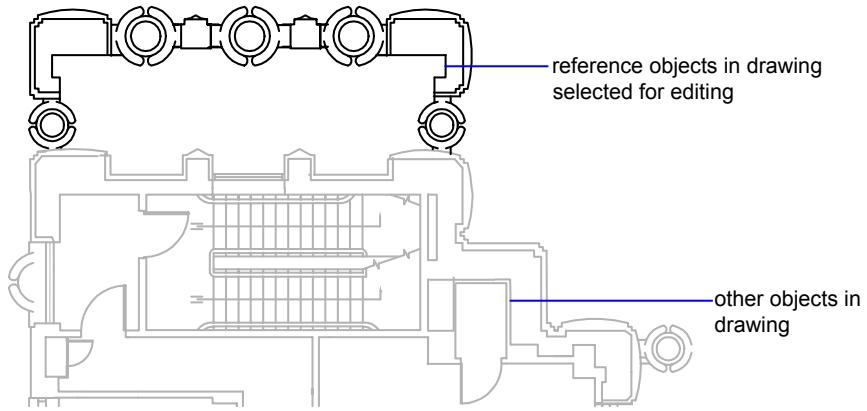
NOTE If you plan to make *major* changes to a reference, open the reference drawing and edit directly within the file. Using in-place reference editing to make major changes can increase the size of your current drawing file significantly during the in-place reference editing session.

Understand the Working Set

Only the objects that are a part of the selected reference are available for selection in the drawing area. The objects you select from the selected reference are extracted and made available for modification in the current drawing. The set of extracted objects, or the *working set*, can be modified and then saved back to update the xref or block definition. Objects that make up the working set are visually distinct from other objects in the drawing. All objects in the current drawing, except objects in the working set, are faded.

Control the Fading of Objects

The XFADECTL system variable controls how objects are displayed while a reference is edited in place. The set of objects extracted from the reference are displayed normally. All other objects in the drawing, including objects in the current drawing and in any references not belonging to the working set, are faded. The value indicates the intensity of display for objects not in the working set. The larger the value is for XFADECTL, the more the objects are faded.



NOTE Objects outside the working set are not faded during in-place reference editing unless SHADEMODE is set to a value of 2D wireframe.

Use the Refedit Toolbar

The Refedit toolbar is displayed and activated after you select which nested objects to edit. Using the buttons on the Refedit toolbar, you can add objects to or remove objects from the working set, and you can discard or save back changes to the reference. Unless it is docked, the Refedit toolbar is automatically dismissed after you save back or discard changes made to the working set.

To edit an xref or block reference in place

- 1 Click Tools menu > Xref and Block In-Place Editing > Edit Reference In-Place.
- 2 From within the current drawing, select the reference that you would like to edit.
If the object you select in the reference belongs to any nested references, all the references available for selection are displayed in the Reference Edit dialog box.
- 3 In the Reference Edit dialog box, select the specific reference that you want to edit.
The reference file is locked to prevent multiple users from opening the file at once. You cannot edit a reference in place if the drawing file is being used by another user.

- 4 Click OK.
- 5 Select the objects you would like to edit in the reference and press ENTER.
The objects that you select become the working set. By default, all other objects are locked and faded.
- 6 Edit the objects in the working set. Click Save Back Changes to Reference.
The objects in the working set are saved to the reference and the xref or the block is updated.

Command line: REFEDIT

Use the Working Set to Edit Referenced Drawings and Blocks

To edit a referenced drawing from within the current drawing, you use the *working set* to identify objects that belong to the xref or block definition rather than the current drawing.

While editing a reference in place, you can add or remove objects from the working set. If you create a new object while editing a reference in place, it is almost always added to the working set automatically. Objects that are not added to the working set are displayed as faded in the drawing.

If a new object is created because of changes made to objects outside the working set, the new object is not added to the working set. For example, your drawing contains two lines that are not a part of the working set. If you edit the lines by using FILLET, a new arc is created between the two lines. The arc is not added to the working set.

When a reference object is part of the working set, you can select the object for editing even if it is drawn on a locked layer in the reference file. You can unlock the object's layer and make changes to the object. Changes made to the object can be saved, but the layer state remains the same in the reference file, whether it is locked or unlocked.

An object that is removed from the working set is added to the host drawing and removed from the reference when changes are saved back. An object that is added to the working set is removed from the host drawing and is added to the reference when changes are saved back. If you create or delete objects, they are automatically added to or removed from the working set. For example, if you use ERASE while editing a reference in place, the erased objects are removed from the working set. You can tell whether an object is in the working set or not by the way it is displayed; a faded object is not in the working set.

The Refedit toolbar is displayed when a reference is being edited in place. The name of the selected reference is displayed in the toolbar. The editing buttons

on the toolbar (Add Objects to the Working Set, Remove Objects from the Working Set, Discard Changes to Reference, and Save Back Changes to Reference) are active only during in-place reference editing. The Edit Block or Xref button is active whenever the toolbar is initiated and a reference editing session is not already in progress within the current drawing. The Refedit toolbar is dismissed automatically after changes made to the reference are saved back or discarded.



To add objects to the working set

- 1 Click Tools menu ► Xref and Block In-Place Editing ► Add to Working Set, or at the Command prompt, enter **refset**.
- 2 If you entered **refset** at the Command prompt, click the Add Objects to the Working Set button on the Refedit toolbar.



- 3 Select the objects you want to add. You can also set PICKFIRST to 1 and create a selection set before using the Add option.
REFSET can be used only with objects in the space (paper space or model space) in which REFEDIT is initiated.

To remove objects from the working set

- 1 Click Tools menu ► Xref and Block In-Place Editing ► Remove from Working Set, or at the Command prompt, enter **refset**.
- 2 If you entered **refset** at the Command prompt, click the Remove Objects from the Working Set button on the Refedit toolbar.



- 3 Select the objects you want to remove. You can also set PICKFIRST to 1 and create a selection set before using the Remove option.
REFSET can be used only with objects in the space (paper space or model space) in which REFEDIT is initiated.

Save Back Edited Referenced Drawings and Blocks

While editing a block reference in place, you either can *save back* or discard changes made to the reference. If you save back changes to a reference, the drawing is regenerated.

When the changes are saved back, the block definition is redefined and all instances of the block are regenerated to reflect the changes. If you choose to discard the changes, the working set is deleted and the block reference returns to its original state.

Similarly, while editing an xref in place, you can save back or discard changes. Objects in the working set that inherit properties not originally defined in the xref retain those new properties. For example, an xref contains layers A, B, and C, and the drawing that references it contains layer D. If new objects are drawn on layer D during in-place reference editing and changes are saved back to the reference, layer D is copied to the xref drawing.

If you remove objects from the working set and save changes, the objects are removed from the reference and added to the current drawing. Any changes you make to objects in the current drawing (not in the xref or block) are not discarded. If you delete any object that is not in the working set, the object is not restored even if you choose to discard changes. You can return the drawing to its original state by using UNDO. If you make unwanted changes to an xref and use REFCLOSE to save back the changes, you must use UNDO to undo any changes made during the reference editing session. After you have undone any unwanted changes, use REFCLOSE to save changes to restore the xref file to its original state.

WARNING While editing a reference in place, if you delete an object that is not in the working set, the object is not restored if you discard changes at the closing of the reference editing session.

Objects in the current drawing that inherit properties defined by the xref retain those new properties. Properties taken from the xref drawing are bound to the current drawing. The xref layer named SITE, for example, appears in the current drawing as \$\$SITE when assigned to an object not in the working set. If BINDTYPE is set to 0, a prefix of \$\$ is added to the reference name in the current drawing. If BINDTYPE is set to 1, reference names remain unchanged in the current drawing, similar to names of inserted objects.

NOTE When you edit and save an xref in place, the original drawing preview is no longer available unless you open and save the referenced drawing.

To save back changes from edited xrefs and blocks

- Click Tools menu ► Xref and Block In-Place Editing ► Save Reference Edits or click the Save Back Changes to Reference button on the Refedit toolbar.

Refedit toolbar



Command line: REFCLOSE

To discard all changes from edited xrefs and blocks

- On the Refedit toolbar, click the Discard Changes to Reference button.

Refedit toolbar



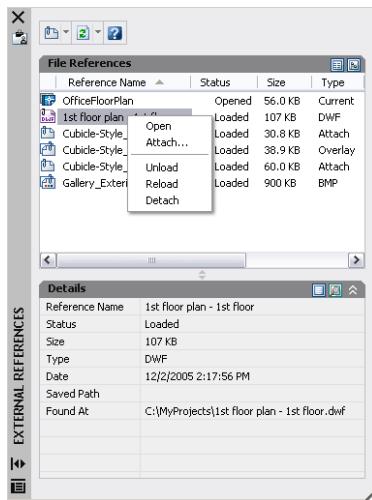
Command line: REFCLOSE

Shortcut menu: With no objects selected during in-place reference editing, right-click in the drawing area. Click Close Refedit Session.

Edit Referenced Drawings and Blocks with Nesting, OLE, or Attributes

There are limitations with editing referenced drawings and blocks from your current drawing if they contain nested objects.

If the reference you select for editing has attached xrefs or block definitions, the reference and its nested references are displayed and available for selection in the Reference Edit dialog box. Nested references are displayed only if the object chosen for selection is part of a nested reference. Only one reference at a time can be selected for editing. If you are editing a reference that contains OLE objects, the OLE objects are displayed but cannot be selected for editing.



If a block reference with attributes is selected for editing, you can choose to display the attribute definitions in the reference and make them available for editing. The attributes are made invisible and the attribute definitions are available for editing along with the selected reference geometry. When changes are saved back to the block reference, the attributes of the original reference remain unchanged. The new or altered attribute definitions only affect subsequent insertions of the block; the attributes in existing block instances are not affected.

Resolve Referenced Drawing Errors

If a referenced drawing cannot be loaded when you open a drawing, an error message is displayed.

Resolve Missing External References

If a referenced drawing cannot be located when you open a drawing, an error message is displayed. There are several options available to you if this occurs.

The program stores the path of the drawing that is used to create the referenced drawing. Each time you load or plot the drawing or use the xref Reload option in the External References palette to update the xref, the program checks this path to determine the name and location of the referenced drawing file. If the name or location of the drawing file has changed, the program cannot reload the xref.

If the program cannot load an xref when it is loading your drawing, it displays an error message. In this example, the program can't find the xref HOUSE:

```
"\acad\dwg\house.dwg": Can't open file
```

```
** Error resolving Xref HOUSE.
```

For each insertion of the referenced drawing, the program displays text (at the location, scale, and rotation angle of the original reference) that contains the path of the missing xref. You can use the XREF Path option to update or correct the path.

One way to avoid these errors is to be sure that when you give other people files that have xrefs attached, you also give them all the referenced files.

Change Nested Xref Paths

When the drawing is reopened and the nested xref is loaded, the program attempts to find the xref in the original xref path first. If the xref is not found, the following search is initiated in the order shown:

- Current folder of the host drawing
- Project search paths defined on the Files tab in the Options dialog box and in the PROJECTNAME system variable
- Support search paths defined on the Files tab in the Options dialog box
- Start In folder specified in the Microsoft® Windows® application shortcut

This ensures that revisions made to the xref are reflected in the current drawing and also makes it possible for the xref to be found if its path has changed.

For example, suppose that the xref tree of the current drawing A is A>B>C, and the owner of drawing B changes the path of xref C to point to *C1.dwg*. When drawing A is reopened, it reflects the path change in drawing B and displays *C1.dwg*. However, if *C1.dwg* is not found, the program looks for xref C at the last location it was saved in drawing A.

See also:

["Update Referenced Drawing Attachments" on page 1090](#)

To change an xref path

- 1 Click Insert menu ▶ External References.
- 2 In the External References palette, select the external reference.
- 3 Under Found At, do one of the following:
 - Edit the xref path directly.

- Click the [...] button and browse to the xref in its new path.

Reference toolbar



Command line: EXTERNALREFERENCES

Resolve Circular External References

A reference file that contains a sequence of nested references that refers back to itself is considered a circular reference. For example, if drawing A attaches drawing B, which attaches drawing C, which attaches drawing A, the reference sequence A>B>C>A is a circular reference.

If the program detects a circular reference while attaching an xref, a warning is displayed asking you if you want to continue. If you respond with yes, the program reads in the xref and any nested xrefs to the point where it detects the circularity. If you respond with no, the process is halted and the xref is not attached.

If a circular reference is encountered while loading a drawing, an error message is displayed and the circular reference for the current session is broken. For example, if you have the circular reference A>B>C>A, and you open *a.dwg*, the program detects and breaks the circularity between *c.dwg* and *a.dwg*. The following error message is displayed:

Breaking circular reference from C to current drawing.

Resolve Name Conflicts in External References

When you attach an xref, the names of its blocks, dimensioning styles, layers, linetypes, and text styles are differentiated from those in the current drawing.

A typical xref definition includes objects, such as lines or arcs. It also includes xref-dependent definitions of blocks, dimension styles, layers, linetypes, and text styles. When you attach an xref, the program differentiates the names of these xref-dependent named objects from those in the current drawing by preceding their names with the name of the referenced drawing and a vertical bar character (|). For example, in the Layer Properties Manager, the xref-dependent named object that is a layer named STEEL in a referenced drawing called *stair.dwg* is listed as STAIR|STEEL.

When you attach an xref, the definitions of its dependent named objects are not added to your drawing permanently. Instead, these definitions are loaded from the referenced drawing file each time you reload it.

Bind Xref-Dependent Definitions

An xref-dependent named object's definition can change if the referenced drawing file is modified. For example, a layer name from a referenced drawing can change if the referenced drawing is modified. The layer name can even disappear if it is purged from the referenced drawing. This is why the program does not allow you to use an xref-dependent layer or other named object directly. For example, you cannot insert an xref-dependent block or make an xref-dependent layer the current layer and begin creating new objects on it.

To avoid the restrictions on xref-dependent named objects, you can bind them to your current drawing. Binding makes the xref-dependent named objects that you select become a permanent part of your current drawing.

When xref-dependent named objects are merged into a drawing through binding, you can use them the same way you use the drawing's own named objects. After you bind an xref-dependent named object, the vertical bar character (|) is removed from the name and replaced with two dollar signs (\$\$) separated by a number (usually zero): for example, the referenced layer, STAIR|STEEL, becomes STAIR\$0\$STEEL. You can then use the RENAME command to change STAIR\$0\$STEEL to STEEL.

If you specify a layer whose associated linetype is not CONTINUOUS, the referenced linetype is also bound. If you apply XBIND to a block, all named objects that are referenced by the objects in the block are also bound. If the block contains a reference to an xref, that xref and all of its dependent definitions are bound.

To bind xref-dependent named objects to the current drawing

- 1 Click Modify menu > Object > External Reference > Bind.
- 2 In the Xbind dialog box, click the plus (+) sign next to an external reference.
The five types of named object definitions (Block, Dimstyle, Layer, Linetype, and Textstyle) are listed.
- 3 Click the plus (+) sign of one of the definition types.
The names of the definition table entries are listed.
- 4 Select a named object definition. Click Add.
The named object definition is listed under Definitions to Bind.
- 5 If necessary, repeat steps 3 and 4.

6 Click OK.

Command line: XBIND

To change the names of layers, dimension styles, and other named objects

- 1** Click Format menu ► Rename.
- 2** In the Rename dialog box, select the named object type and then the item you want to rename.
- 3** Enter the new name in the Rename To box below the old name.
- 4** Click Rename To. Click OK.

Command line: RENAME

Track External Reference Operations (Log File)

The program can maintain a log of its actions while attaching, detaching, and reloading xrefs and while loading a drawing containing xrefs. This log is maintained only if the XREFCTL system variable is set to 1. The default setting is 0.

The log file is an ordinary ASCII text file with the same name as the current drawing and the file extension *.xlg*. If you load a drawing with the file name *sample.dwg*, for example, the program searches for a log file named *sample.xlg* in the current folder. If the file does not exist, a new file is created with that name.

Once a log file has been created for a drawing, the program continues to append information to it. The program writes a title block to the log file each time the file is opened. If the log file becomes too large, you can delete it.

Example: A Sample Title Block from an Xref Log File

This title block contains the name of the current drawing, the date and time, and the operation being performed.

```
=====
Drawing: detail
Date/Time: 09/28/99 10:45:20
Operation: Attach Xref
=====
```

During a detaching or reloading operation, the program prints the nesting level of all affected xrefs immediately following the title block. To see a

reference tree for a set of xrefs in your current drawing, use Detach or Reload and check the resulting entries in the log file.

Example: A Sample Log File Entry Showing Nested Xrefs

In the following example, the xref ENTRY_DR contains two nested xrefs: HARDWARE and PANELS. The xrefs HARDWARE and PANELS also each contain two xrefs.

```
=====
Drawing: detail
Date/Time: 10/05/99 15:47:39
Operation: Reload Xref
=====
Reference tree for ENTRY_DR:
ENTRY_DR Xref
-HARDWARE Xref
--LOCKSET Xref
--HINGES Xref
-PANELS Xref
--UPPER Xref
--LOWER Xref
```

The program writes an entry in the log file for each xref-dependent named object temporarily added to the current drawing and for any errors that occur. Most error messages are written both to the screen and to the log file.

Example: A Sample Log File That Shows the Results of Attaching an Xref

The following example shows a partial listing of the log file entries generated when the external reference STAIR is attached to the working drawing *test.dwg*. The log file lists the definition (symbol) table affected and the name of the definition added, along with a status message.

```
=====
Drawing: test
Date/Time: 12/18/99 14:06:34
Operation: Attach Xref
=====
Attach Xref STAIR: \ACAD\DWGS\STAIR.dwg
Searching in ACAD search path
Update block symbol table:
Appending symbol: STAIR|BOLT
Appending symbol: STAIR|BOLT-HALF
...
block update complete.
Update Ltype symbol table:
Appending symbol: STAIR|DASHED
Appending symbol: STAIR|CENTER
Appending symbol: STAIR|PHANTOM
Ltype update complete.
Update Layer symbol table:
Appending symbol: STAIR|STEEL-HIDDEN
```

```
Appending symbol: STAIR|OAK
...
Layer update complete.
STAIR loaded.
```

To use the xref log file

- 1 At the Command prompt, enter **xrefctl**.
- 2 Enter **1** to turn logging on or **0** to turn logging off.
- 3 Press ENTER.
Logging is off by default.

Increase Performance with Large Referenced Drawings

There are several features that can improve performance when dealing with large referenced drawings.

Overview of Demand Loading

Provides a high-level description of using demand loading to improve performance when working with large referenced drawings.

The program uses *demand loading* and saving drawings with indexes to increase performance with large referenced drawings that have been clipped using the program or that have many objects on frozen layers. With demand loading, the program loads into memory only the data from the reference drawing that is necessary to regenerate the current drawing. In other words, referenced material is read in “on demand.” Demand loading works in conjunction with the INDEXCTL, XLOADCTL, and XLOADPATH system variables.

Unload Xrefs

When an xref is unloaded from the current drawing, the drawing opens much faster and uses less memory. The xref definition is unloaded from the drawing file, but the pointer to the reference drawing remains. The xref is not displayed, and nongraphical object information does not appear in the drawing. However, you can restore all the information by reloading the xref. If XLOADCTL (demand loading) is set to 1, unloading the drawing unlocks the original file.

You should unload a reference file if it is not needed in the current drawing session but may be used later for plotting. You can maintain a working list of unloaded xrefs in the drawing file that you can load as needed.

To unload an xref

- 1 Click Insert menu ► External References.
- 2 In the External References palette, select the reference name you want to unload.
- 3 Right-click, and click Unload.

Reference toolbar



Command line: EXTERNALREFERENCES

Work with Demand Loading

With demand loading, only the data from the referenced drawing that is necessary to regenerate the current drawing is loaded into memory.

To realize the maximum benefits of demand loading, you need to save the referenced drawings with layer and spatial indexes. The performance benefits of demand loading are most noticeable when you do one of the following:

- Clip the xref with the program to display a small fraction of it. A spatial index is saved in the externally referenced drawing.
- Freeze several layers of the xref. The externally referenced drawing is saved with a layer index.

If demand loading is turned on, and you have clipped xrefs that were saved with spatial indexes, objects in the referenced drawing database that are contained within the clip volume comprise the majority of the objects read into the drawing. If the clip volume is modified, more objects are loaded as required from the reference drawing. Similarly, if you have xrefs with many layers frozen that were saved with layer indexes, only the objects on those thawed layers are read into the current drawing. If those xref-dependent layers are thawed, the program reads in that geometry from the reference drawing as required.

When demand loading is turned on, the program places a lock on all reference drawings so that it can read in any geometry it needs to on demand. Other users can open those reference drawings, but they cannot save changes to

them. If you want other users to be able to modify an xref that is being demand loaded into another drawing, use demand loading with the Copy option.

If you turn on demand loading with the Enable with Copy option, the program makes a temporary copy of the referenced drawing and demand loads the temporary file. You can then demand load the xref while allowing the original reference drawing to be available for modification. When you turn off demand loading, the program reads in the entire reference drawing regardless of layer visibility or clip instances.

Layer and spatial indexes were added in AutoCAD Release 14 and AutoCAD LT 97. If you externally reference a drawing saved in a release previous to this, you do not see the same performance benefit as drawings saved with the indexes. For maximum performance, use demand loading with referenced drawings saved with layer and spatial indexes turned on in AutoCAD Release 14, AutoCAD LT 97, or more recent versions.

To turn on demand loading

- 1 Click Tools menu ▶ Options.
- 2 In the Options dialog box, Open and Save tab, under External References, select Disabled, Enabled, or Enabled with Copy.
- 3 Click OK.

Work with Layer and Spatial Indexes

To receive the maximum benefit of demand loading, it is recommended that you save any drawings that are used as xrefs with layer and spatial indexes.

A layer index is a list showing which objects are on which layers. This list is used when the program is referencing the drawing in conjunction with demand loading to determine which objects need to be read in and displayed. Objects on frozen layers in a referenced drawing are not read in if the referenced drawing has a layer index and is being demand loaded.

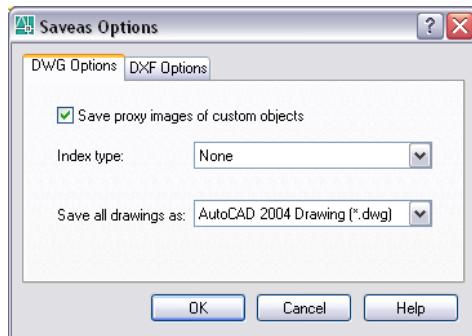
The spatial index organizes objects based on their location in 3D space. This organization is used to efficiently determine which objects need to be read in when the drawing is being demand loaded and clipped as an xref. If demand loading is turned on, and the drawing is attached as an xref and clipped, the program uses the spatial index in the externally referenced drawing to determine which objects lie within the clip boundary. The program then reads only those objects into the current session.

Spatial and layer indexes are best used in drawings that will be used as xrefs in other drawings where demand loading is enabled. Drawings that are not

going to be used as xrefs will not benefit from layer and spatial indexing or demand loading.

To save a drawing with layer and spatial indexes

- 1 Click File menu ► Save As.
- 2 In the Save Drawing As dialog box, click Tools menu ► Options.



- 3 In the Saveas Options dialog box, under Index Type, select Layer, Spatial, or Layer & Spatial. Click OK.

NOTE If you are saving a partially open drawing that does not already contain layer and spatial indexes, this option is disabled.

- 4 Click Save.

Set Paths for Temporary Xref File Copies

When you turn on demand loading with copy, the XLOADPATH system variable can be used to indicate the path where copies of externally referenced drawings are to be placed. The path you specify remains in effect for all drawing sessions until you indicate a different path. If no value for XLOADPATH is specified, the temporary file copies are placed in the standard folder for temporary files.

If you find that referencing drawings over a network is slow, it is recommended that you set XLOADPATH to reference a local folder, and set XLOADCTL to 2 so that the externally referenced files are demand loaded from your local machine. Conversely, to minimize the number of temporary files created by multiple users referencing the same drawing, those users can set XLOADPATH to point to a common folder. In this manner, multiple sessions of the program can share the same temporary copies of reference drawings.

You can set XLOADPATH in the Options dialog box and indicate the path where copies of externally referenced files are to be placed.

To set the file path for xref copies

- 1 Click Tools menu ► Options.
- 2 In the Options dialog box, Files tab, select the *Temporary External Reference File Location* folder.

The expanded tree view displays the path where copies of xref files are placed. If no path is specified, the temporary copies are placed in the location specified by Temporary Drawing File Location.
- 3 Edit the path by selecting it and entering a new path.
- 4 Click OK (or Apply) to set the path.

Part 10

Create Realistic Images and Graphics

Chapter 33 Add Lighting to Your Model

33

Add Lighting to Your Model

Lighting can be added to help you visualize your model while you work or to prepare your model for rendering.

In this chapter

- Overview of Lighting
- Create and Manipulate Lights
- Simulate Sunlight

Overview of Lighting

Good lighting is crucial for visualizing a shaded 3D model as you work and for creating a rendering.

Default Lighting

When you work in a viewport with a 3D shaded view, the default lighting is 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. Default lighting must be turned off in order to display lighting from user-created lights or the sun.

User-Created Lights

For more control over lighting, 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. The effects of changes are visible in the viewport immediately.

Spotlights and point lights are each represented by a different light glyph. Distant lights and the sun are not represented by glyphs in the drawing. You can turn display of light glyphs on or off while you work. By default, light glyphs are not plotted.

The Sun

The sun is a special light similar to a distant light. The angle of the sun is defined by the geographic location that you specify for the model and by the date and time of day that you specify. You can change the intensity of the sun and the color of its light.

To convert lights created in previous releases to lights in AutoCAD 2007 format

Create and Manipulate Lights

You can add point lights, spotlights, and distant lights and set the location and properties of each.

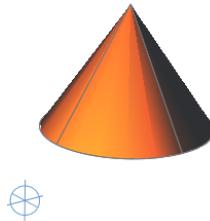
Choose the Type of Light to Use

Each type of light has a different effect in the drawing.

You can use a command to create a light, or you can use a button on the Lights toolbar or the Lights panel in the dashboard. Use the Properties palette to change the color of a selected light or its 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.

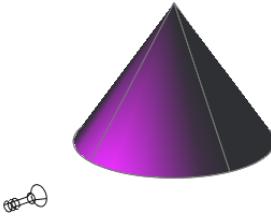
Point Lights

A point light radiates light in all directions from its location. The intensity of a point light diminishes over distance unless attenuation is set to None. Use point lights for general lighting effects.



Spotlights

A spotlight emits a directional cone of light. You can control the direction of the light and the size of the cone. The intensity of spotlights diminishes over distance. Spotlights are useful for highlighting specific features and areas in your model.



Distant Lights

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. There is no light glyph to represent a distant light in your drawing.

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 a backdrop uniformly.

- 1 Click View menu » Render » Light » New Point Light.
- 2 Click in the drawing to specify a location for the light.
- 3 Enter **n** and enter a name.
- 4 Press ENTER to exit the command.

Select the light and use grip tools and the Properties palette to change its properties.

Lights toolbar



Command line: POINTLIGHT

Lights panel (click to expand), Create a Point Light

- 1 Click View menu » Render » Light » New Spotlight.
- 2 Click in the drawing to specify a location for the light.

- 3 Click to specify a target.
- 4 Enter **n** and enter a name.
- 5 Press ENTER to exit the command.

Select the light and use grip tools and the Properties palette to change its properties.

Lights toolbar



Command line: SPOTLIGHT

Lights panel (click to expand), Create a Spotlight

- 1 Click View menu » Render » Light » New Distant Light.
- 2 Click in the drawing to specify a location for the light.
- 3 Click to specify a direction.
- 4 Enter **n** and enter a name.
- 5 Press ENTER to exit the command.

Enter **lightlist** to open the Lights in Model window. Select the light and use the Properties palette to change its color and other properties.

Lights toolbar



Command line: DISTANTLIGHT

Lights panel (click to expand), Create a Distant Light

- 1 Click Tools menu » Palettes » Tool Palettes.
- 2 Select a point light or a spotlight whose properties you want to use for the tool.
- 3 Drag the light to the active tool palette.

The new light tool retains all of the properties of the light except its location properties.

To control the appearance of light glyphs

- 1 Click Tools menu » Options » Drafting tab » Lights Glyph Settings.

- 2 In the Light Glyph Appearance dialog box, click Point or Spot to display a point light or spotlight glyph in Glyph Preview.
- 3 To change colors, click Edit Glyph Colors to open the Drawing Window Colors dialog box.
For more information, see To customize the colors of the application window elements
- 4 To change the size of the glyph in the drawing, use the Glyph Size slider.
- 5 Click OK to exit each dialog box.

Command line: OPTIONS

To show or hide light glyphs

- 1 Click Tools menu » Palettes » Dashboard.
 - 2 On the Light panel, click the arrow to expand.
 - 3 Click Light Glyphs On to show light glyphs or Light Glyphs Off to hide them.
- The display setting does not control plotting of light glyphs.

Command line: LIGHTGLYPHDISPLAY

To plot light glyphs

- 1 Click Tools menu » Palettes » Properties.
- 2 Select a point light or a spotlight.
- 3 In the Properties palette, change the Plot Glyph property to Yes.

Standard toolbar



Command line: PROPERTIES

Control the Location and Properties of Lights

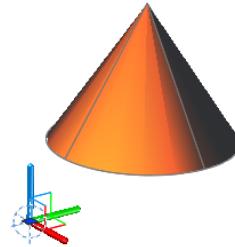
Each light you add to the drawing is listed by name and type in the Lights in Model window (LIGHTLIST). Lights in blocks and xrefs, the sun, and default lighting are not included.

When a light is selected in the list, it is selected in the drawing and vice versa. The properties of the lights in the list are saved per drawing. All properties can be changed in the Properties palette. When a light is selected in the drawing, you can use grip tools to move or rotate the light and change some other properties, for example, the hotspot and falloff cone in spotlights. You can see the effect on the model as you change the properties of a light.

The following properties are common to all lights in the list:

- **Status**. Controls whether the light emits light when the default lighting is turned 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**. Sets a multiplier that controls brightness. Intensity is not related to attenuation.
- **Color**. Sets the color of the light emitted.

You can use the grip labeled Position to move a point light or a spotlight, or you can set the location in the Properties palette. The Position grip moves the light but does not change the target. To move both the light and its target, drag the light glyph itself.



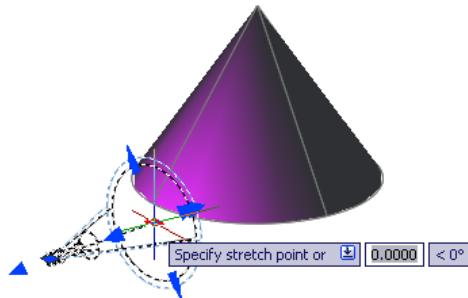
A distant light requires a direction, which you set by specifying a source vector. You can use the pointing device to specify a FROM point and a TO point, or you can enter a direction vector.

Attenuation (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).

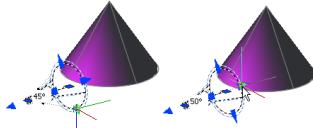
Another way to control the start point and end point of light is to use limits. Limits work like clipping planes to control where light is first emitted and where it stops. The use of limits can increase performance by removing the need for the program to calculate light levels where the light is already practically invisible.

NOTE Attenuation limits are supported only for rendering, not in viewports. The OpenGL driver (*wopengl9.hdi*) does not support attenuation start limits and end limits. To identify your driver, enter **3dconfig**, and click Manual Tune. Look at the selected Driver Name in the Manual Performance Tuning dialog box.



Hotspot and Falloff (Spotlights)

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.

To select a light

To delete a light

To set attenuation in a point light or a spotlight

- 1 Click Tools menu » Palettes » Properties.
- 2 Click the light glyph, or select the light in the Lights in Model window.
- 3 In the Properties palette under Attenuation, set Use Limits to Yes, or select a type of attenuation:
 - **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.

- 4 If you set Use Limits to Yes, enter the following values in the properties palette or use the grip tool in the drawing to set the limits:

■ **Start Limit Offset.** Specifies the point where light starts as an offset from the center of the light. The default is 0.

■ **End Limit Offset.** Specifies the point where light ends as an offset from the center of the light. No light is cast beyond this point.

NOTE Attenuation start limits and end limits are not supported by the OpenGL driver (*wopengl9.hdi*). To identify your driver, enter **3dconfig**, and click View Tune Log. Look at the Driver line in the Performance Tuner Log.

To specify a source vector for a distant light

- 1 Click View menu » Render » Light » Light List.
- 2 In the Lights in Model window, select the distant light.
- 3 Click Tools menu » Palettes » Properties.
- 4 In the Properties palette under Geometry, use one of the following methods to change a value:
 - Enter a value in a cell.
 - Click the Calculator button to use QuickCalc.
 - Click the Select in Drawing button and specify a From point and a To point with the pointing device. The line displayed in the drawing represents the vector.

Lights toolbar



Command line: LIGHTLIST

Lights panel, Light List

To move a point light or a spotlight

- 1 Select the light glyph, or select the light in the Lights in Model window.
- 2 Select the Position grip.

The Position grip is at the bottom of a spotlight and at the center of a point light.

- 3 Drag the light to the new location and click to drop it there.

NOTE A spotlight rotates as it moves to maintain its aim at the target. If you want to move both source and target, drag the light glyph itself, not the Position grip.

To change the color of a light

- 1 Click Tools menu » Palettes » Properties.
- 2 Select the light glyph, or select the light in the Lights in Model window.
- 3 In the Properties palette, click in the Color cell and select a color. Click Select Color to open the Select Color Dialog Box.

Command line: PROPERTIES

Simulate Sunlight

The sun is a light that simulates the effect of sunlight and can be used to show how the shadows cast by a structure affect the surrounding area.

The rays of the sun are parallel and have the same intensity at any distance. Shadows can be on or off. To improve performance, turn off shadows when you don't need them. All settings for the sun except geographic location are saved per viewport, not per drawing. Geographic location is saved per drawing.

The angle of the light from the sun is controlled by the geographic location you specify for your model and by the date and time of day. These are properties of the sun and can be changed in the Sun Properties Window and the Geographic Location Dialog Box. The time zone used is based on the location, but you can adjust it independently (TIMEZONE system variable).

To turn the sun on and off in a drawing

- 1 Click View menu » Render » Light » Sun Properties.
- 2 In the General settings, click the Status setting, and select On or Off.

Lights toolbar



Command line: SUNPROPERTIES

Light panel, Sun Status

To change the brightness of the sun

- 1 Click View menu » Render » Light » Sun Properties.
- 2 In the General settings, click the Intensity Factor setting, and select a new value.

Lights toolbar



Command line: SUNPROPERTIES

Light panel (click to expand), Brightness slider (affects all lights in the viewport)

To change the geographic location of your model

- 1 Click View menu » Render » Light » Geographic Location.
- 2 In the Geographic Location dialog box, use one of the following methods:
 - In Region, select a region. Select the Nearest Big City check box. Click a location on the interactive map.
 - In Region, select a region. In Nearest City, select a city.
 - Enter a latitude and its direction and a longitude and its direction.
- 3 View the time zone, and if it is not correct, select the correct time zone from the list.
- 4 In North Direction, click the interactive image or enter a value to specify the angle of north on the XY plane of the WCS in your drawing. The diagram updates to show the current setting.
- 5 Click OK.

Lights toolbar



Command line: GEOGRAPHICLOCATION

Light panel (click to expand), Geographic Location

To change the angle of the sun

- 1 Click View menu » Render » Light » Sun Properties.
- 2 In the Sun Angle Calculator settings, click the Date setting, and select a new date.
- 3 Click the Time setting, and select a new time.

Use the Daylight Savings setting if necessary.

Lights toolbar



Command line: SUNPROPERTIES

Light panel, Date slider and Time slider

To change the color of the sun

- 1 Click View menu » Render » Light » Sun Properties.
- 2 In the General settings, click the Color setting, and select a color.
Click Select Color to open the Select Color Dialog Box.

Lights toolbar



Command line: SUNPROPERTIES

Lights panel, Edit the Sun

Index

- ^ (caret) 814
 - stacked text indicator 814
- (hyphen) 823
 - no value in field display 823
- (hyphen), dialog box suppression character 38
- (minus sign), DesignCenter tree view level display button 47
- PLOT command 977
- , (comma) 871
 - formula argument delimiter 871
- ? (question mark) 296, 844
 - wild-card character 296, 844
- . (period) 296, 844
 - wild-card character 296, 844
- .dws extension for standards files 144
- .dxx file name extension 571
- (L) least material condition symbol 938
- { } (braces, open and close) 851
 - format codes for 851
- @ (at sign) 296, 331–332, 335, 337–338, 652, 844
 - break point without a gap symbol 652
 - relative coordinate specifier 331–332, 335, 337–338
 - wild-card character 296, 844
- * (asterisk) 361, 844
 - prefix in pointer input, accessing 361
 - wild-card character 844
- * (asterisk), wild-card character 296
- / (forward slash), stacked text indicator 814
- /b command line switch 74
- /c command line switch 74
- /ld command line switch 74
- /nologo command line switch 75
- /nossm command line switch 75
- /p command line switch 75
- /r command line switch 75
- /s command line switch 74
- /set command line switch 75
- /t command line switch 74
- /v command line switch 74
- /w command line switch 75
- # (pound sign) 295, 359, 361, 814, 823, 844, 868
 - absolute coordinates entry in pointer input 359
 - formula error display 868
 - invalid field display 823
 - prefix in pointer input, accessing 361
 - stacked text indicator 814
 - wild-card character 295, 844
- + (plus sign) 47, 383–384
 - DesignCenter tree view level display button 47
 - tracking points marker 383–384
- = (equal sign) 868, 871
 - formula start character 868, 871
- | (vertical bar) 1106
 - dependent named object name divider 1106
- ~ (tilde) 296, 844, 849
 - incorrect word flag 849
 - wild-card character 296, 844
- \$ (dollar sign) 868, 1106
 - bound named object name dividers 1106
 - formula cell address fix value prefix 868
- \$\$, bound xref name prefix 1101
- 2D Cartesian coordinates (*X*, *Y* coordinates) 275, 328–332, 381–382
 - absolute values 328, 330, 332
 - coordinate values 275
 - entering 330, 332, 381–382
 - with coordinate filters 381–382
 - point locations, displaying 329

- point values, displaying 329
- relative values 328, 331–332
- X, Y* values 330
- 2D coordinates 328, 332
 - polar coordinates 328, 332
- 2D objects 763
 - sectioning 763
- 2D solids 323, 782, 788, 879
 - and associative dimensioning 879
 - creating 782
 - modifying 788
 - simplified display 323
- 2D UCS icon 356
- 2D wireframe visual style 178
- 3D arrays 623, 625–626
 - creating 623, 625–626
- 3D blocks 471
 - Block Editor and 471
- 3D Cartesian coordinates (*X, Y, Z* coordinates) 175, 275, 329, 334–336, 381–382
 - absolute values 335–336
 - defining 3D views with 175
 - displaying coordinate values 275
 - entering 334–336, 381–382
 - by digitizing 335
 - with coordinate filters 381–382
 - point locations, displaying 329
 - point values, displaying 329
 - relative values 335–336
 - Z* value defaults 335
- 3D coordinates 336–339
 - cylindrical coordinates 336–337
 - spherical coordinates 338–339
- 3D DWF files 1058, 1062–1064
 - exporting 1062
 - publishing 1062–1063
 - publishing objects to 1063
 - viewing and printing published files 1058, 1064
- 3D DWF publishing 1029, 1058
- 3D DWF Publishing 1058
- 3D edges. *See* edges (of 3D solids)
- 3D hidden visual style 178
- 3D isometric views 174
- 3D modeling 30
 - dashboard and 30
- 3D models (images) 178, 184, 189–191, 680, 711–712, 722–723, 761, 1117–1118, 1127–1128
 - cameras and 184
 - creating geographic locations of models 1127–1128
- interactive 3D view
- intersection problems in 711–712
- lighting 1117–1118
- live sectioning 761
- motion path animations 189–190
- overview 680
- performance issues and recording animation 191
- solid modeling 680
- visual styles
- visual styles and 178
- wireframes 722–723
- 3D objects 189, 623, 625, 722–725, 728–730, 761
 - arraying 623, 625
 - clipping
 - hidden lines. *See* hidden lines (in 3D objects)
 - live sectioning 761
 - lofting 729–730
 - motion path animations and 189
 - See also* 3D solids
 - See also* surface meshes
 - See also* wireframes
 - sweeping 728
 - thickness 724–725
 - viewing
 - dynamically
 - wireframes 722–723
- 3D Orbit view
 - interactive 3D view
 - starting
- 3D solid edges 743–744, 746
 - chamfering
 - cleaning up redundant edges 746
 - coloring
 - copying

filleting
imprinting objects on 743–744
moving
rotating
rules and limitations
scaling
selecting
working with

3D solid faces 743–744, 746
cleaning up redundant faces 746
coloring
copying
deleting
imprinting objects on 743–744
moving
rotating
rules and limitations
scaling
selecting
working with

3D solid vertices 746
cleaning up redundant vertices 746
modifying
moving
rotating
rules and limitations
scaling
working with

3D solids 393, 471, 681, 691, 695–699, 701, 704, 706–713, 717, 722–725, 728–734, 743–747, 750
calculating total area 393
chamfering
checking for interferences 711–712
cleaning 746
composite solids 708–709, 731–733
creating 697–699, 701, 704, 706–707, 728–730
by lofting 699, 701, 729–730
by revolving 704, 729–730
by sweeping 697–698, 728
from objects 704, 706
from surfaces 707
overview
cross sections 750
edges on

extruding 695–697, 728
by direction 696
from objects 696, 728
from objects
from paths 695, 697
from tapered angles 696

filleting
history 681
imprinting objects on 743–744
in dynamic blocks 471
lofting 729–730
manipulating 728
meshes 713, 717
polysolids 691, 728
pressing or pulling bounded areas 734
primitives
rules and limitations
sectioning 750
separating into original shapes 744–745
shells 745
slicing 710–711
subobjects. *See* 3D solid edges
subobjects. *See* 3D solid faces
subobjects. *See* 3D solid vertices
thickness 724–725
types of 681, 728
validating 746–747
wireframes 722–723

3D space 343–344, 346, 349, 366, 616, 630–631, 635–637, 645
defining UCSs in 346
drawing planes in 343, 349
specifying 343
dynamic viewing
extending objects in 635–636
filleting objects in 645
mirroring objects in 630–631
relocating UCSs in 344
rotating objects in 616
trimming objects in 635, 637
using object snaps in 366

3D surfaces. *See* surfaces

3D UCS icon 356–357
changing the appearance of 357

3D views 174–177, 189–191, 323, 346, 354, 735, 738, 741
animation settings
architectural vs. mechanical design conventions 175
changing navigation modes
changing to plan views 177
defining 175
dynamic viewing
Fill mode option 323
flying through models
grip tools in 735
interactive 3D view
isometric views 174
motion path animations 189–190
moving objects in 738
multiple viewports in 346, 354
named views
parallel/perspective projection
preset views 174–175
recording animations 191
resetting view
restoring the default view 177
rotating 175
rotating objects in 741
setting 176
 with angles of rotation 176
 with viewpoint coordinate 176
specifying
 interactively
starting
visual styles
walking through models
3D wireframe visual style 178
3DARRAY command 623
3DCORBIT command
3DDISTANCE command
3DDWFPPUBLISH command 1029, 1058
 in network and stand-alone installations 1029
3DFORBIT command
3DMESH command 717, 720
3MOVE command 736
 grip tools and 736
3DORBIT command
 interactive navigation

 starting
3DPAN command
3DROTATE command 736
 grip tools and 736
3DSWIVEL command
3DZOOM command

A

absolute coordinates 328, 330, 332, 334–336, 359, 361
 entering 328, 330, 332, 334–336
 Cartesian 330, 332, 335–336
 polar 332, 334
 entering, in pointer input
 tooltips 361
 specifying in pointer input 359
absolute paths 1082
 for xrefs 1082
absolute value function (in QuickCalc) 400
absolute world coordinate system (WCS)
 coordinates 361
 entering, in pointer input
 tooltips 361
acad.err file 134
acad.exe 76
 command line switches 76
acceleration
 shadow display and
accuracy. *See* precision
ACI colors (numbers) 303–304
 setting (for new objects) 304
Acrobat PDF files 1022
 plotting 1022
action override properties 516–517
 defined 516
 specifying 516–517
action selection sets 496, 513, 518, 550
 assigning actions to 518
 defined 496
 for array actions 513
 including parameters in 513
 modifying 550
actions 242, 605–607
 click actions in tree views 242

for dynamic blocks. *See* actions (for dynamic blocks)
redoing 605–607
undoing. *See* undoing actions
actions (for dynamic blocks) 468, 470, 476–477, 479–482, 484, 496, 498–500, 502–504, 506–508, 510–518, 520, 531
adding to dynamic block
 definitions 468, 470, 482, 496, 498–500, 502–504, 506–508, 510–516, 531
array actions 513–514
flip actions 511–512
lookup actions 515–516, 531
methods 470
move actions 498–499
overview 468, 496
parameters and 482
polar stretch actions 506–507
rotate actions 508, 510
scale actions 500, 502
stretch actions 503–504
assigning to parameters 477, 496, 498–500, 502–504, 506–508, 510–516, 520, 531
array actions 513–514
flip actions 511–512
grips and 520
lookup actions 515–516, 531
methods 477
move actions 498–499
overview 496
polar stretch actions 506–507
rotate actions 508, 510
scale actions 500, 502
stretch actions 503–504
assigning to selection sets 518
 methods 518
Block Authoring Palettes
 window 476
defined 468, 482, 496
deleting from parameters/parameter sets 477
dependencies (dynamic block element relationships) 470, 482
dependency highlighting 479, 481
displaying in Block Editor 479–481
override properties 516–517
parameters supported by 498
selection sets. *See* action selection sets types 482, 484, 496, 498
 overview 496
 by parameter type 482, 484
 parameter types supported by 498
Adaptive Adjustment option (Plot Style Table Editor) 1014
adaptive degradation
 performance and
Adaptive Degradation and Performance Tuning dialog box
adaptive grid display 373
Add-a-Plotter wizard 974
adding 83, 394, 396, 453–454
 areas 394, 396
 regions 453–454
 tool flyouts 83
Adobe Acrobat PDF files 1022
 plotting 1022
Adobe PostScript driver 1024
Advanced Setup wizard 110–111
 starting drawings 111
Aerial View window 164–167
 displaying the entire drawing in 166
 panning with 164–166
 resizing the image in 166–167
 updating the image in 166–167
 updating viewports in 167
 view box 164
 zooming with 164, 166–167
aliases 34
 entering commands with 34
aligned dimensions 876, 909, 911
 creating 911
 extension line origins 909
 See also dimensions
aligning 233–234, 388–391, 618–619, 795, 797, 802–803, 821, 852, 862, 892, 895, 904, 906, 930
 dimension text 892, 895, 930
 with dimension lines 892, 895

lateral tolerances 904, 906
leader text 821
objects 234, 388–391, 618–619
 in different layout
 viewports 234
 marking equal segments on other
 objects 388, 390–391
 marking measured intervals on
 other objects 388–389
text 795, 797, 802–803, 852, 862
 in tables 862
 multiline text 802–803, 852
 single-line text 795, 797
views, in layout viewports 233
alignment grips 520, 561
alignment parameters 482, 484, 492
 actions supported by 482, 484
 adding to dynamic block
 definitions 492
 defined 484
 grip type 482
alignment paths 377
 displaying 377
alignment paths (AutoTrack) 383–384
 in object snap tracking 383–384
alternate units (for dimensions) 898–900
 adding and formatting 900
 conversion values 899
 symbol for 898–899
alternative font 836, 838, 852
 specifying 836, 838, 852
 format code 852
anchor 45
 DesignCenter 45
anchored windows 40, 64, 67, 88
anchoring dashboard 32
AND operator, in Help topic searches 6
ANGDIR system variable 360
 and dimensional input 360
angle brackets 35, 332, 898
 double close, transparent command
 prompt indicator 35
 open and close 898
 primary units symbol 898
 open, polar coordinate
 delimiter 332
primary units symbol 898
angle direction 118, 207, 332
 setting 118
 specifying 207, 332
 for polar coordinates 332
angle offset action override
 property 516–517
 defined 516
 specifying 516–517
angle overrides 380
 operations with precedence
 over 380
 specifying points 380
angles 118, 175–176, 332, 373, 378,
 380, 391–392, 408, 450, 789,
 840, 852, 1129
 bisecting, with construction
 lines 450
 calculating 391–392
 defining 3D views with 175–176
dimensioning. *See* angular dimensions
direction. *See* angle direction
displaying the angle of a line 408
hatch angle 789
locking, specifying points by 380
polar angles 378, 380
for polar coordinates 332
polar. *See* polar angles
snap angle 373
sunlight 1129
surveyor angles 118
text obliquing angle 840, 852
 format code (multiline
 text) 852
units. *See* angular units
angular dimensions 876, 920–921, 932
 creating 920–921
 definition points 932
 See also dimensions
angular unit conventions 118
 setting 118
angular units 118, 207, 328
 setting 118, 207
 types 328
Animation Preview dialog box

animations 69, 189–191
3D view settings
compression 191
exporting 191
for panning and zooming 69
for running scripts 69
motion path animation 189–191
playing
saving 191
annotations 205
scaling 205
ANSI code pages supported by
Windows 894
aperture box (AutoSnap) 368
apostrophe 35
transparent command character 35
appending objects to selection sets 595
application window 62, 64, 66
coloring elements in 62, 66
font selection 64
applications 572
destination. *See* container applications
(destination applications)
opening attribute extraction files in
other applications 572
source. *See* server applications
ARC command 359
dimensional input 359
arc length dimensions 923–924
creating 923–924
uses 923
arc polylines, drawing 420, 422, 438,
440
arc symbol 923
display options 923
architectural blocks, accessing collections
of 58
architectural units 116, 902–903
specifying 116
suppressing zero feet/inch
values 902–903
archiving 261, 1091
drawings with xrefs 1091
sheet sets 261
arcs 189, 432–436, 445, 665–666, 697–
698, 916, 919–920
connecting 434
creating centerlines/center marks
on 916, 919
creating solids from
dimension line arcs 920
dimensioning. *See* radial dimensions
drawing 432–436
by specifying three points 432,
435
tangent to arcs 436
tangent to lines/arcs 434
using start point, center point,
and angle 432
using start point, center point,
and endpoint 432, 435
using start point, center point,
and length 433
using start point, endpoint, and
direction/radius 433
elliptical arcs 445
joining to polylines 665–666
motion path animations and 189
sweeping to create objects 697–698
See also objects
arcs (in revision clouds) 455, 457
editing lengths 457
setting default values for
lengths 457
setting lengths 455
AREA system variable 393
areas 392–394, 396–397
adding 394, 396
calculating 392–393, 396
enclosed. *See* regions
plot area. *See* plot area
subtracting 394, 397
ARG (profile text) files 72
array actions 513–514
adding rotate actions to 514
adding to dynamic block
definitions 513–514
parameters supported by 513
rows and columns 513
specifying 513

selection sets 513
 including parameters in 513

arrays (multiple copies) 622–626
 3D 623, 625–626
 creating 622
 limiting in size 623
 polar 623–624, 626
 rectangular 623

arrow grips 755

arrowheads 877, 887–888
 choosing 887
 control options 887
 customizing 888
 flipped arrowhead appearance in earlier versions 887
 flipping 888
 scale factors 888

arrows. *See* arrowheads

ARX applications 74
 loading with /ld command line switch 74

Asian-language fonts. *See* Big Fonts

Associated Standards File(s) icon 148

associating 144, 934–935
 dimensions 934–935
 standards files with drawings 144

association points (of dimensions) 934

associative dimensions 878–879, 931, 935
 associativity. See associativity (of dimensions)
 converting into nonassociative dimensions 935
 converting nonassociative dimensions into 935
 as disassociated automatically 931
 filtering selections 878
 modifying 931
 object types supported/not supported 879
 updating 879

associative hatches 664, 772, 788
 exploding 664

associativity (of dimensions) 878–879, 932, 934–935
 changing 879, 934–935
 the default 879
 determining 878
 partial 932
 system variable controlling 879
 types 878

asterisk (*) 361, 844
 prefix in pointer input, accessing 361
 wild-card character 844

asterisk (*), wild-card character 296

at sign (@) 296, 331–332, 335, 337–338, 652, 844
 break point without a gap symbol 652
 relative coordinate specifier 331–332, 335, 337–338
 wild-card character 296, 844

attached labels
 See also labels

attached xrefs 1079, 1083, 1090–1091
 notification of 1079, 1083
 relocated xrefs 1083
 updating 1090–1091
 with demand loading turned on 1090

attaching 563, 565, 1079–1081, 1088
 attributes to blocks 563, 565
 xrefs 1079–1081, 1088
 with DesignCenter 1079, 1081
 overlaying xrefs vs. 1088
 tools for 1079
 See also linking

attenuation of light 1124–1125
 point and spotlights 1124–1125
 See also falloff angle (of spotlights)

attribute definitions 564–566, 1103
 changing the order of 566
 creating 564–565
 editing 564, 566
 in-place reference editing and 1103

attribute extraction files 570–572
 creating 570
 error handling in 572

formats 570
nested blocks in 572
opening in other applications 572
See also attribute extraction template files
attribute extraction table 567–568
 updating 567–568
attribute extraction template files 568–570, 572–573
 aligning columns 573
 attribute tag fields in 569
 creating 568, 572
 elements in 568
 numeric field format codes 570
 sample file 570
Attribute Extraction wizard 566
attribute prompts 565
 order of 565
attribute tags 564, 569, 576, 578
 defined 564
 duplicate tags 576, 578
 highlighting 576, 578
 fields in attribute extraction template files 569
attribute values 576, 578
 changing the prompt order 576, 578
 See also attribute information
attributes (of blocks) 563–568, 573–574, 576–577, 579–580, 1028, 1103
attaching to blocks 563, 565
characteristics 564
defining 564–565
displayable. *See* labels
exporting. *See* attributes (of blocks), extracting information from
extracting information from 564, 568, 573
in-place reference editing and 1103
invisible 564
modifying 576–577, 580
 applying changes to block references 577
 in block definitions 576–577
 in block references 577, 580
outputting to files 567
overview 563
querying 566–567
redefining 574
removing 576, 579
selecting 565
specifying for publishing in DWF files 1028
stand-alone 565
updating 576–577
uses 563
variable. *See* variable attributes
visibility 564
ATTSYNC command 577
AUDIT command 134
auditing drawings 134, 142, 146–147, 149, 152
 examples 134
 for standards violations 142, 146–147, 149, 152
 multiple drawings 147, 152
authenticity of drawing files 122
 checking 122
Auto-hide option 64, 68, 88–89
Auto-list 806
Autodesk Authorized Training Centers 12
Autodesk DWF Composer 1019
Autodesk DWF Viewer 758, 1019, 1027–1028, 1050
 section objects and 758
 viewing DWF files in 1050
 viewing published files 1027
Autodesk e-learning lessons 12
Autodesk Educational Products 1080
Autodesk Express Viewer. *See* Autodesk DWF Viewer
Autodesk Official Training Courseware 12
Autodesk products 10, 20
 information resources
 late-breaking information about 10, 20
 support. *See* product support
Autodesk Subscription 13
Autodesk Vault 15–16
 overview 15
 Vault client software 15

Vault Data Management Server 15
Vault Office Add-in 16
Autodesk website 10
AutoLISP Compatibility Analyzer (Migration Tool) 2
Autolist. *See* Auto-list
automatic saves 129, 136
 specifying 129, 136
AutoSnap 365, 368
 running object snap display options 368
 settings 368
 tools 368
AutoStack 815
AutoStack Properties dialog box 816
AutoTrack 383, 385
 settings 385
averaging the values in table cell ranges 869
axes 339, 344–345, 350–351, 511, 618, 702, 738, 740, 742
 constraining movement to 738, 740
 constraining rotation to 742
 defining for UCSs 350–351
 positive direction of 344
 rotation about 344
 reflection lines 511
 revolving objects around 702
 rotating objects around 618
 rotating UCSs around 345
 in the WCS 339
axis of revolution/rotation 730
 relocating 730

B

back, moving objects to 325
background colors 62, 790, 798, 862, 872, 1065
 for drawings 62
 for DWF plot files 1065
 masking objects with 790
 multiline text object fills 798
 for the table cell and row indicator display 872
tables 862

background lines. *See* hidden lines (in 3D objects)
background plotting 977–978
 canceling 978
 turning on/off 977
background publishing 978, 1028, 1030
 canceling 978
 canceling published jobs in process 1030
 checking published-job-in-process status 1030
 turning on/off 1030
backgrounds 798
 multiline text object options 798
 visual styles and
backup files 129, 136–137, 244
 file name extension 136
 restoring drawings from 136–137
 saving 129, 136
 when saving drawings 129, 136
 for sheet set data (DST) files 244
BAK file name extension 136
balloon messages, in Communication Center 18
balloon notification, in Communication Center 18–19
 turning off 19
base grips 655, 755
base parameters 482, 484, 496
 actions supported by 482, 484
 adding to dynamic block definitions 496
 grip type 482
base points 373, 461, 465, 471, 501–502, 509
 dependent base points 501, 509
 dynamic block base point 471
 for inserting blocks 461, 465
 independent base points 501–502, 509
 snap base point 373
Base Type property 501, 509
 for rotate actions 509
 for scale actions 501
baseline dimensions 876, 909, 912, 930
 creating 912

dimension line spacing 930
See also dimensions

basic dimensions 903, 905

basic math functions (in QuickCalc) 400
using 400

batch audit reports 147, 152–153
adding notes to 152
exporting 153
filtering data 152
generating 152
viewing 152
See also Batch Standards Checker

batch audits 147, 152
running 147, 152
See also batch audit reports

batch plot lists 1049

BP3 files 1049

batch plot. *See* BP3 files

Batch Standards Checker 147, 150
starting 150
See also batch audits

beam angle (spotlights) 1124

below ground. *See* underground (3D modeling)

beveling objects. *See* chamfering

Big Fonts 834–836
assigning to text styles 836
font file 834
specifying 835

bills of materials 566

binding 1091–1092, 1106
dependent named objects to drawings 1106
xrefs to drawings 1091–1092

BINDTYPE system variable 1101

bisecting 450
angles, with construction lines 450

bitmaps (BMP images) 610
mapping onto 3D object surfaces. *See* image mapping
See also BMP (bitmap) files

blips 607–608
removing 607–608

BLK files. *See* block template (BLK) files

block attribute information 566–568
extracting 567

output to file 567
updating in table 567–568
turning off notification for 568

block attributes. *See* attributes (of blocks)

Block Authoring palettes 471, 476–477

block authoring tools 476
customizing 476

BLOCK command 127, 129–130
saving block definitions 127, 130
saving selected objects 127, 129

block definition tables 460

block definitions 50, 53, 127, 130, 460–464, 466, 472–473, 549, 551–552, 573–577, 583–585, 1095–1096
copying 464
creating 461–462, 473
in Block Editor 473
creating drawing files from 466
displaying properties
dynamic. *See* dynamic block definitions
libraries of 463–464
modifying 573, 583, 1095–1096
modifying 574
modifying attributes in 576–577
opening 472
opening 549
removing 584–585
saving 551–552
saving as drawing files 127, 130
tables 460
updating 50, 53, 574–575

block descriptions 574–575
modifying 574–575

Block Editor 468, 471–481, 549, 551–552
3D blocks and 471
actions as displayed in 479–481
specifying text color 480
specifying text size 480
updating text size 481

Block Authoring palettes in 477

closing 475

commands-not-allowed-in message 471

creating block definitions 473

defined 468

dependency highlighting 479, 481
dimensions in. *See* parameters (for dynamic blocks)
displaying block definition properties 475
displaying object properties 475
drawing area 471
grips as displayed in 479–480
invisible objects as displayed in 481
modifying dynamic block definitions 549
opening block definitions 472, 549
opening blocks 473–474
opening drawing files as blocks 474
opening dynamic blocks 474
parameters as displayed in 478–481 settings overview 478 specifying color 479 specifying font 480 specifying text and arrow size 480 updating text size 481 preventing access to 476 saving block definitions 551–552 toolbar 471 UCS command in 471 Block Editor drawing area 471 block instances. *See* blocks block instances. *See* blocks (block references)
block libraries (symbol libraries) 463–464 creating 463–464 block library drawings (symbol library drawings) 558, 574 inserting block definitions from 558 updating block definitions from 574 block placeholder fields, in block attribute definitions 823 block references blocks as. *See also* blocks block references. *See* blocks (block references)
block template (BLK) files 1050, 1055–1057 creating 1056
editing 1057
including information in published DWF files 1055
block tools 80, 92, 557 prompt for rotation angle 80, 557 refreshing image on tool palette 92 block-related metadata options for published DWF files 1050 block-related properties and attributes 1028 specifying for publishing in DWF files 1028
blocks 122, 459, 601, 663, 665, 822, 1093, 1095, 1102, 1106 attributes. *See* attributes (of blocks) binding referenced objects 1106 clipped 1093 clipping 1095 descriptions. *See* block descriptions drawing files as 122 inserting 122 editing. *See also* in-place reference editing exploding 665 grips in 663 vs. groups 601 inserting drawing files as 122 inserting. *See also* inserting blocks modifying *See also* in-place reference editing nested. *See also* nested blocks OLE objects in 1102 pasted text as 822 selection sets of objects within. *See* working sets blocks (block properties) 557 drawing files as 557 inserting 557 blocks (block references) 47, 50, 54–55, 58, 80–81, 85, 205, 207, 459–467, 473–474, 553, 555–559, 563–565, 573–577, 579–581, 583–584, 764–765, 828, 864, 867, 879, 1054, 1096, 1098, 1101–1102 accessing 47, 54 online 54

adding to tool palettes 50
attaching attributes to 563, 565
base points 461, 465
colors in 553, 581
contextual fields in 828
copying 464–465
 block definitions 464
 paper space objects between drawings 465
creating 460–464, 466, 556
 block references 556
 drawing files as 466
 drawing files as 464
 in block libraries 463–464
 methods 460–462
creating tools for 80
defined 459, 461
defining 461–462, 564
discarding changes to 1102
downloading 55, 58
drawing files as 464, 466, 474, 557, 559, 574–575
 creating 464, 466
 inserting 557, 559
 opening 474
 updating block definitions from 574–575
dynamic. *See* dynamic blocks
editing in place 1096, 1098
exploding 583–584
flattened views of 3D objects 765
including block information in
 published DWF files 1054
inserting 467, 556–557, 864, 867
 methods 556
 in table cells 864, 867
 with tool palettes 467, 557
linetypes in 553, 581
lineweights 553
modifying 573, 577, 580
 block attributes in 577, 580
nested blocks 555
object property insert settings 553
opening 473–474
organizing 467
placing 81, 558
placing from tool palette 85
 prompt for rotation angle 85
redefined 879
as references in drawings 556
saving back changes to 1101–1102
scaling 205, 207
scaling automatically 81, 558
sectioning 764
updating 465, 576, 579–580
using a tool created from 85
 vs. xrefs 465
BMP (bitmap) files
 See also bitmaps (BMP images)
boldfacing multiline text 804
Boolean operators 6
 Help topic searches 6
borders (of table cells) 861, 863
boundaries 169, 220–221, 223, 452, 454
 creating 452, 454
 regions out of 452, 454
of layout viewports 220–221, 223
 clipping 221, 223
 suppressing display in nonrectangular viewports 220
of views 169
boundary edges 632
boundary points 779–780
 defining hatch boundaries with 779–780
boundary polylines, creating 421, 423, 439, 441
boundary sets 421, 423, 439, 441
 for boundary polylines 421, 423, 439, 441
boundary sets (for hatch areas) 775, 778
 defining 778
bounded areas of 3D solids 734
bounding boxes (for multiline text) 798–799
 drawing 798–799
boxes 164, 681–682, 728, 821
 3D solids 681
 as primitive solids 728
 creating 682
 text boxes for leader text 821

view box (Aerial View window) 164
See also pickbox cursor

BP3 files. *See* batch plot lists

braces ({}) 851
format codes for 851

BREAK command 652

breaking 652, 654
objects 652, 654

brightness 1128
sunlight 1128

bringing objects to front 325
draw order of objects 325

broken pencil icon 357

browsing 57
for online content 57

bulleted lists 805–807, 809–810
bullets for 805
creating as you type 805, 809
formatting multiline text as 806
moving items down a level 810
removing list formatting from 807
separating items from 810

bullets 805
for bulleted lists 805

BYBLOCK setting 279, 304, 312, 321, 553
color value 304
linetype value 312
lineweight value 321

BYLAYER setting 273, 279, 304–305, 312–313, 321–322, 553
color value 304–305
linetype value 312–313
lineweight value 321–322

C

CAD Manager Control utility 56
accessing 56
installing 56
used to turn DC Online tab on/off 56

CAD Standards Settings dialog box 143, 148

CAD standards. *See* standards (for named objects)

CALCINPUT system variable 403

calculating 206, 391–393, 396, 403, 414
angles 391–392
areas 392–393, 396
circumferences 393
distances 391–392
grid limits 206
mathematical expressions in dialog boxes 403, 414
perimeters 393
point coordinate values 392

calculator variables (in QuickCalc) 410–413
creating 411, 413
deleting 413
displaying information on 410
editing 413
loading into the Input box 410
organizing 412
predefined variables 412
types 411

calculators. *See* QuickCalc calculator

calligraphy pen-style revision clouds (markups) 455–456

callout blocks (for sheet views) 250–251, 256–258
adding 258
avoiding prompts when placing 251
changing 256–257

Camera Preview dialog box 185

Camera Properties palette

cameras 184–191, 758
about 184
clipping planes 184
creating 184–185
displaying
glyphs 188
lens length 187
linking to paths 189
locations 186, 188
motion path animations and 189–190
moving 188
naming 187
properties 185
recording animations and 191

rotating
section objects and 758
targets 188
canceling 35, 606, 1030
commands 35, 606
published jobs in process 1030
caret (^), stacked text indicator 814
Cartesian coordinates 328, 361
entering in dynamic input
 tooltips 361
catalogs (of parts) (online) 54, 56–57
 accessing 54, 56–57
CDF (comma-delimited format) files 571
cell address format 868
 formula cell address fix value
 prefix 868
cell and row indicator display (in
 tables) 872
 background color control 872
 turning off/on 872
cell properties 866
 changing/restoring defaults 866
 copying to other cells 866
cell ranges 868–871
 averaging the values in 869
 counting the cells in 870
 format 868, 871
 summing the values in 869
cells (in tables) 825, 864–872
 address format 868
 border properties 866
 copying formulas to other cells 868
 copying properties to other cells 866
 counting the cells in ranges 870
 editing text 864, 867
 entering formulas 868, 871
 entering text 864–865
 formatting text 864–865
formulas in 868–872
 copying to other cells 868
 entering 868, 871
 inserting 868–870
 inserting fields 872
inserting blocks 864, 867
inserting fields in formulas 872
inserting fields in text 825, 868
 inserting formulas 868–870
 line breaks in 865
 moving the cursor within/between
 cells 865
 properties 866
 border properties 866
 changing/restoring defaults 866
 copying to other cells 866
 ranges. *See* cell ranges
selecting text 865
text in 825, 864–865, 867–868
 editing 864, 867
 entering 864–865
 formatting 864–865
 inserting fields 825, 868
 line breaks 865
 selecting 865
cells (tables) 856, 858–859, 862–863
 gridlines 863
 margins 862
 merging 858–859
 modifying 856, 858
CELTSCALE system variable 315, 430
center marks 877, 916, 919
 control options 916
 creating 916, 919
centerlines 878, 916, 919
 control options 916
 creating 916, 919
 automatically with radial
 dimensions 919
centimeters 114–115, 117
 converting inches to 114, 117
 converting to inches 115, 117
certification of graphics hardware and
 software
Chain Actions property 484, 539, 547–
 548
chain dimensions. *See* continued
 dimensions
CHAMFER command 647
chamfer distances 648, 650
 overriding 648
 setting 648, 650
chamfering 647–652
 3D solid edges

3D solids
and trimming 649
creating chamfers 647
distance setting for 648, 650
hatch boundaries 647
line segments 650
multiple sets of objects 647, 652
objects 647–651
 by setting the distances 648, 650
 by specifying length and angle 649, 651
 without trimming 651
polyline segments 649
polylines 649, 651
chamfers
 creating. *See* chamfering
channels, in Communication Center 18
character formatting, in multiline text 804
character spacing (multiline text), format code 852
characters 815
 stacking 815
Check Standards dialog box 147
checking 15, 122, 142, 146, 149, 152, 746–747, 848–849, 1030
 3D solids 746–747
 drawing files 122
 for authenticity 122
 drawings 142, 146, 149, 152
 for standards violations 142, 146, 149, 152
 files in and out 15
published-job-in-process status 1030
spelling
 in another language 848
 text not checked 848
 See also dictionaries
Checking Complete alert summary
 (standards checking) 147
CHECKSTANDARDS command 147
chords (in revision clouds) 457
 editing lengths 457
CIRCLE command 359
 dimensional input 359
circles 189, 393, 436–437, 916, 919
 calculating
 areas/circumferences 393
 creating centerlines/center marks
 on 916, 919
 dimensioning. *See* radial dimensions
 drawing 436–437
 tangent to objects 436–437
 motion path animations and 189
 See also objects
circular references (between xrefs) 1105
 resolving 1105
circumferences, calculating 393
circumscribed polygons 424
 drawing 424
classification property, selecting objects by 594
Clean Screen (View menu) 62
cleaning up 3D solids 746
cleaning up the screen (display) 607
clearing the screen 62
click actions in tree views 242
Clipboard (Windows) 608–611
 copying objects to 609–610
 cutting objects to 608–610
 pasting objects from 610–611
clipdepth (of clipping boundaries) 1094
clipped references 1093
clipping 1095, 1110
 references 1095, 1110
 xrefs 1110
clipping boundaries (for blocks and xrefs) 1079, 1093–1094
 clipdepth 1094
 displaying 1079
 turning off 1093
 types 1093
clipping planes 184
 3D objects
 cameras and 184
 setting (adjusting)
closed polylines 420, 438
 drawing 420, 438
closing 25, 38, 245
 sheet sets 245
 Text window 38

toolbars 25
clouds. *See* revision clouds (markups)
codes. *See* control codes
collaborative environments, maintaining standards in 142, 146–147
collections (of content), specifying 55, 58
colon 868, 871
 cell range character 868, 871
color attribute of materials. *See* diffuse color (of materials)
color book colors 308
 searching for swatches in color books 308
color books 303, 305–309
 browsing 308
 changing colors 306–307
 layer colors 306
 object colors 307
 file path 308–309
 changing 308
 defining multiple folders for 309
 installing 303, 307–308
 searching for color swatches in 308
 setting colors (for new objects) 305, 307
Color control (Properties toolbar) 273, 279
color depth 1067
 for DWF plot files 1067
 setting 1067
color mapping tables 1007–1008
 deleting 1007–1008
color names 303
color numbers. *See* ACI numbers
color palettes 303
color swatches 308
 searching for, in color books 308
color-dependent plot style tables 975, 996–997, 1001, 1007
 and color mapping tables 1007
 converting 997
 to named plot style tables 997
 converting drawings from named tables to 997
 converting drawings to named tables from 996–997
 file name extension 975
 plot styles in 996
 predefined tables 1001
color-dependent plot styles 975, 996–997
 converting drawings from named styles to 997
 converting drawings to named styles from 996–997
coloring 62, 66, 362, 799, 804, 851, 866
 application window elements 62, 66
 multiline text 799, 804, 851
 format code 851
objects. *See* object colors
table cell borders 866
 tooltips 362
 See also colors
colors 271, 273, 279, 286, 291, 302–306, 371, 479, 553–554, 562, 581–582, 594, 610, 657, 757, 862–863, 1009–1011, 1123, 1127, 1129
assigning 273, 279, 286, 302, 304
 to layers 279, 286
 to objects 273, 302, 304
 to 3D solid subobjects
background. *See* background color
block parameters 479
changing 291, 305–306, 581–582
 for layers 581
 layer colors 291, 305–306
 object colors 305, 582
converting to grayscale 1010–1011
copying 610
current color 303
custom grips, changing 562
dithering 1009, 1011
filtering selection sets based on 594
for 3D faces
grid color 371
for grips 657
in blocks 553–554, 581
light colors 1123, 1127, 1129
for objects. *See* object colors

section plane indicator 757
setting (for new objects) 273, 302,
 304
setting (making current) 554
table background 862
table borders or gridlines 863
table text 862
uses 302
visual styles and
See also coloring
See also layer colors
columns (in array actions) 513
 offset 513
columns (in tables) 857–862, 869–870
 adding 859
 averaging values in 869
 counting cells in 870
 deleting 860
 formatting 861–862
 header rows 862
 modifying 857–859
 See also database table columns
 setting number of 857
 summing the values in 869
combining 452, 454
 regions 452, 454
comma (,) 871
 formula argument delimiter 871
comma-delimited format (CDF) files 571
comma-separated format (CSV) files 860
command aliases 34
 entering commands with 34
command history 37
 displaying all 37
command line 34, 36, 38, 40–41
 copying text from the text window
 to 38
 displaying 38
 prompts when entering
 commands 38
 displaying when hidden 40–41
 entering commands on 34
 entering system variables on 36
 hiding 40–41
 prompts on, displaying when entering
 commands 38
switching between dialog boxes
 and 38
command line calculator 414
 overview 414
command line prompts 62
 coloring 62
command line switches 74–76
 precedence of environment
 settings 76
 starting the program by using 74,
 76
 syntax 75–76
command line window. *See* command
 window
command options, specifying 34
command prompts. *See* command line
 prompts
command prompts. *See* dynamic prompts
Command Reference, accessing 10
Command shortcut menu 27, 37
 displaying 27
command strings, creating command tools
 for executing 87
command tools 86–87
 AutoLISP routines, VBA
 macros/applications,
 executing 86
 creating 86–87
 creating/using 86
 executing command strings,
 customized commands, or
 scripts 86
 using 87
command window 36–37, 39–41, 66,
 358
 anchoring 40
 displaying when hidden 40–41
 docking 40–41
 and Dynamic Input 358
 editing in 36
 floating (undocking) 39–40
 font selection 66
 hiding 40–41
 navigating in 36
 repeating commands in 37
 resizing 37, 40–41

switching to the text window
from 37

transparency setting 40

commands 10, 27, 34–35, 37–38, 86–87,
224, 358, 402–403, 471, 606

accessing QuickCalc from
within 402–403

accessing recently used
commands 37

cancelling 35, 606

commands-not-allowed-in-Block-Editor
message 471

completing with Dynamic
Input 358

copying command already used 35

creating tools for executing 86–87

customized 87

- creating command tools for
executing 87
- creating command tools from
CUI dialog box 87

cycling through commands on
command line 34

editing commands. *See* editing
commands

entering 34, 38

- on the command line 34
- displaying prompts on the
command line 38
- suppressing dialog boxes 38

executing 34

- by right-clicking 34

for viewing 224

- scale locking and 224

Help on 10

interrupting with other
commands/system
variables 35

prompts. *See* command line prompts

prompts. *See* dynamic prompts

repeating 27, 34, 37

- in the command window 37
- by right-clicking 27

restarting recently used
command 34–35

right-clicking in the drawing area
while in/not in progress 27

specifying command options 34

transparent commands 35

Commands tab (Help window) 8

information depth 8

comments

- on drawings. *See* markups

Communication Center 17–19

- balloon notification 18
- channels 18
- customizing 19
- frequency of updates 18
- information displayed in 18
- opening 17, 19
- privacy and 17
- turning off balloon notification 19

comparing dimension styles 881–882

complete plotter configuration files. *See*
PCP (legacy plotter configuration)
files

composite 3D solids 731–734, 744–745

- history 731–733
- manipulating 733
- selecting individual solids in 734
- selecting original subobjects
- separating into original shapes 744–
745
- working with 731

composite regions, creating 452, 454

composite solids 708–709, 731–734,
744–745

- creating 708–709
- defined
- editing 733
- history 731–733
- selecting individual solids 734
- selecting original subobjects
- separating into original shapes 744–
745
- working with 731

composite tolerances 939

compressing text 795

compression (video) 191

Concepts tab (Help window) 8

information types 8

Conceptual visual style 178
cones 681, 684–686, 728
 3D solids 681
 as primitive solids 728
 creating 684–686
 solid cone frustum 685
conical helixes 448
connectivity feature. *See* database connectivity feature
constants
 calculator variables. *See* global constants
constrained orbit in 3D Orbit view
constraining object movement 738, 740–741
constraining object rotation 742
construction lines (xlines) 234, 449–451
 aligning objects in different layout
 viewports with 234
 bisecting angles with 450
 drawing 450–451
 offsetting 450
construction plane. *See* XY plane (construction plane)
content (for drawings)
 (DesignCenter) 47–51, 54–58
 accessing 47, 54, 56–57
 online 54, 56–57
 adding 49
 bookmarking 47
 changing the source displayed 48
 collections of, specifying 55, 58
 creating tool palettes with 50–51
 online content 54–58
 accessing 54, 56–57
 browsing 57
 downloading 55, 58
 previewing 55, 58
 searching for 54, 57–58
 types 56
 shortcut folder. *See* Favorites folder
content area (DesignCenter Online) 54, 57
 controlling the number of categories/items displayed in 57
content area (DesignCenter) 44, 49, 51
 displaying detail in 49
 loading 51
Contents tab (Help window) 3–4
 displaying (viewing) topics 4
 expanding/contracting the list 4
context-sensitive Help 10
 F1 key 10
 See also Quick Help (Info palette)
contextual fields (in text) 828
 listing of 828
continued dimensions 876, 909, 912–913, 930
 creating 912–913
 dimension line spacing 930
 See also dimensions
CONTINUOUS linetype 278
continuous orbit in 3D Orbit view
contours of lofted objects 730
control grips (in blocks) 663
control panel icons 31
control points 670–671
conversions (of units) 410
 copying to the QuickCalc Input box 410
converting 114–115, 117, 153–154, 409–410, 423, 446, 610–611, 704–706, 837, 899–900, 935, 996–997, 1010–1011
associative/nonassociative dimensions 935
colors 1010–1011
 to grayscale 1010–1011
degrees to radians 410
dimension units (primary to alternate) 899–900
drawing units 114–115, 117
drawings to a different plot style type 996–997
fonts 837
layers to drawing standards 153–154
objects to solids or surfaces 704–706
older versions of lights
pasted information to drawing file
 format 610–611
polylines into lines 423

radians to degrees 410
spline-fit polylines into splines 446
units of measurement 409
See also migrating
CONVTOSOLID command 705–706
CONVTOSURFACE command 704–705
Coons surface patch meshes 716, 721
coordinate display (cursor location) 29,
 328, 330, 359
 changing 330
 in tooltips 359
 types 328
coordinate filters 381–382
 specifying points with 381–382
coordinate systems 328, 339, 430
 Cartesian 328
 mapping, on paper to the WCS 430
 origin 328, 339
coordinate systems, in model space
 viewports 195
coordinate values 359, 361
 entering 361
 in dynamic input tooltips 361
 entering in tooltips 359
coordinates 29, 186, 188, 275, 328–332,
 336–339, 345, 357, 359, 392, 408
absolute values 328, 330, 332
camera locations 186, 188
Cartesian 328
cursor coordinate display 29, 359
 in tooltips 359
cursor location display 328
cylindrical 336–337
displaying coordinate values 275,
 408
entering
 broken pencil icon warning 357
 relative to the WCS 345
 relative values 328, 331–332
 See also under specific types of
 coordinates
filters. *See* coordinate filters
point locations, displaying 329
point values 329, 392
 calculating 392
 displaying 329
polar coordinates 328, 332
relative values 328, 331–332
spherical coordinates 338–339
unit types 328
COPYHIST command 37
copying 35, 37, 101, 126, 215, 276, 279,
 286, 288, 464–465, 477, 609–610,
 620–622, 660–662, 866, 868,
 1005, 1032, 1035, 1039, 1042,
 1111–1112
3D solid subobjects
block definitions 464
cell properties to other table
 cells 866
colors 610
command already used 35
drawing sheets (in drawing
 sets) 1032, 1035, 1039,
 1042
between drawings 126
formulas to other table cells 868
layers, between drawings 288
layouts 215
named plot styles 1005
object properties 276
 suppressing specific
 properties 276
objects 279, 609–610, 620–622,
 660–662
 in arrays 622
 to the Clipboard 609–610
 with grips 660–661
 multiple copies 621, 660–661
 with offset snaps 660, 662
 point method 620–621
 by relative displacement 621
 with rotation snaps 660, 662
 to other layers 279
paper space objects, between
 drawings 465
parameter sets 477
properties 286
 layers 286
text, all of, in the text window 37
tool palettes between tool palette
 groups 101

xrefs, for demand loading 1111–1112
correcting drawings. *See* markups
correcting dynamic block
definitions 550–551
correcting mistakes. *See* undoing actions
correcting spelling. *See* checking spelling
counting the cells in table cell
ranges 870
covering objects (with blank areas) 790
CPolygons (crossing polygon
selections) 591–592
Create Layout wizard 213, 215, 965
Create New Drawing dialog box 108–109
displaying 108
starting drawings in 108–109
Create New Table Style dialog box 861
Create Sheet Set wizard 242
Create Support Request option,
Subscription Center 13
cross (+), UCS icon symbol 356
cross sectioning 750, 761, 763
2D and 3D sections 763
3D objects 750
live sectioning 761
See also sectioning
cross sections 700–701, 729–730
changing contours of 730
lofting objects with 700–701, 729–730
crosshairs 62, 359
coloring 62
coordinate display 359
in tooltips 359
crosshatching. *See* hatching
crossing polygon selections 591–592
crossing selections 590
crossing windows. *See* crossing selections
CSV (comma-separated format) files 860
CTB file name extension 975
CTRL+A keys, toggling group
selection 602
CTRL+click, cycling through objects 589
CTRL+ENTER keys 865
saving table changes 865
CTRL+R keys 196–197
cycling through viewports 196–197
cubic feet units (in QuickCalc) 404
Current Object Scale value (linetype
scale) 315
cursor 29, 328, 359, 376, 386, 865
coordinate display 29, 328, 359
in tooltips 359
crosshairs. *See* crosshairs
moving from table cell to table
cell 865
pickbox. *See* pickbox cursor
rubber-band line 376, 386
cursor menus
See also shortcut menus
curve-fit polylines, creating 666
curves. *See* arcs
custom dictionaries (spelling
checker) 848–850
creating 850
editing 848–850
custom grips (dynamic blocks) 561–562
changing color 561
changing display color 562
manipulating dynamic blocks
with 562
types 561
alignment 561
flip 561
linear 561
lookup 561
rotation 561
standard 561
table of 561
custom grips (parameters). *See* grips (for
parameters) (custom grips)
custom hatch patterns 773
defining 773
custom menus 131
specifying 131
custom paper sizes 951–953, 981, 983
adding 951–952, 981, 983
deleting 953
editing 952, 983

- custom properties 259–260, 468, 470, 539–540, 561–562
 - in dynamic blocks 468, 470, 539–540, 561–562
 - displaying/not displaying 539–540
 - extracting 539
 - manipulating blocks with 470, 561–562
 - overview 468
- for sheets/sheet sets 259–260
 - creating 260
- custom scales 987
 - plotting using 987
- Custom tab (Drawing Properties dialog box) 120
- customization (CUI) files 25
 - specifying 25
- customized commands 87
 - creating command tools for executing 87
 - creating command tools from CUI dialog box 87
- customizing 26–28, 31, 61, 72, 84, 367, 476, 596, 888
 - arrowheads 888
 - dashboard 31
 - object selection 596
 - object snap menu 367
 - parameter sets 476
 - program environment 61, 72
 - with profiles 72
 - right-clicking 26, 28
 - shortcut menus 26–27
 - tool flyouts 84
 - workspaces
- cutaway geometry 762
- cutaway views (of 3D objects), creating
- cutting 608–610, 710–711
 - 3D solids 710–711
 - objects to the Clipboard 608–610
- cutting edges 632
- cutting planes 757
 - cross sections
 - properties 757
- cycling 196–197, 368, 588–589, 602
 - through objects 589, 602
 - in groups 602
- through running object snaps 368
- through selected objects 588
- through selected subobjects
- through viewports 196–197
- Cycling property 521
- cylinders 681, 686–687, 728
 - 3D solids 681
 - as primitive solids 728
 - creating 686–687
- cylindrical coordinates, specifying a point with 336–337
- cylindrical helixes 448

D

- damaged drawing files 134–135
 - repairing 134–135
- dashboard 30–32
 - anchoring 32
 - autohiding 32
 - control panels in 30
 - customizing 31
 - displaying 32
 - overview 30
 - slide-out panels 32
 - tool palette groups 32
- data labels. *See labels*
- data sources
 - See also* external databases
 - data types in tables 861–864
- database information (for objects), listing 275
- database records
 - editing
 - See also* database table data
 - fields, data labels. *See labels (data labels)*
 - viewing
 - in filtered subsets. *See* queries
- database table data
 - See also* database records
- database tables
 - columns. *See* database table columns

rows. *See* database records
See also database table data

databases

- external. *See* external databases

datum (for geometric tolerances) 936, 938

- reference frames 938
- reference letters 936, 938

datum (for ordinate dimensions) 922

- See also* ordinate dimensions

DBX applications 74

- loading with /ld command line
- switch 74

DC Online tab (DesignCenter) 47, 56

DCTCUST system variable 849

DCTMAIN system variable 849

DDEDIT command 842

decurving polylines 667

default drawing folder 122–123

- changing 122–123

default drawing template file 109–110, 112–113

- restoring 112–113
- starting drawings with 109–110

Default shortcut menu 27

- displaying 27

default visual styles 178, 181

default workspaces 75

- loading with command line
- switch 75

definition points (of dimensions) 932

definition table items 237

- See also* named objects
- eliminating unnecessary items 237

defpoints. *See* definition points (of dimensions)

degrees 410

- converting radians to 410
- converting to radians 410

deleting 95, 100, 170, 198, 215, 263–265, 283, 288–289, 301, 310, 343, 413, 477, 527, 545, 550, 604, 832, 860, 953, 969, 1006–1008, 1086

3D solid subobjects

3D solid vertices

actions from parameters/parameter sets 477

calculator variables 413

color mapping tables 1007–1008

columns/rows in tables 860

custom paper sizes 953

dynamic block definition

- elements 550

layer settings 301

layers 283, 288–289

layouts 215

lights

linetypes 310

named groups 604

named page setups 969

named plot styles 1006–1007

named views 170

search paths 1086

sheet selections 263–265

text styles 832

tool palette groups 100

tool palettes 95

UCSs 343

value set values 545

viewport arrangements 198

visibility states 527

delimiter characters, in multiline text lists 806

DELOBJ system variable 455

demand loading (of xrefs) 1090, 1110–1112

- copying xrefs for 1111–1112
- saving xrefs with indexes 1110–1111
- turning on/off 1111
- updating xrefs when turned on 1090

DEMANDLOAD system variable 828

dependencies (dynamic block element relationships) 470, 551

- overview 470
- viewing 551

dependency highlighting 479, 481

- in Block Editor 479
- turning on/off 481

dependent base points 501, 509

dependent named objects (in xrefs) 1105–1107
binding to drawings 1106
listing 1107
name conflicts 1105
referencing, as impossible 1106
See also named objects
dependent symbols
See also dependent named objects (in xrefs)
desaturating colors
design environment. *See* model space
design standards. *See* standards, for named objects
Design Web Format files. *See* DWF (Design Web Format) files
DesignCenter 44–45, 47–50, 52–53, 119, 463–464, 474, 558–559, 574, 773
accessing content 47
adding content 49
adding items to tool palettes 50
anchored 45
content area 49
 displaying detail in 49
content specification tabs 47
copying block definitions 464, 574
dragging hatch patterns into drawings 773
Home button folder, changing 48
home folder 52
 setting to a folder with symbol library drawings 52
inserting blocks 558–559
loading content area with hatch patterns 52
loading content area with symbol libraries 52
opening blocks in Block Editor from 474
opening drawings from 53
toolbar 45
updating block definitions 50, 53
uses 44, 119
viewing block descriptions 463
DesignCenter Online 54–57
 content area 54
content specification views 54
content types 56
privacy 55
retrieving content on the Web 57
turning on/off using CAD Manager Control utility 56
DesignCenter window 44–46, 96, 237–238, 1079, 1081
 attaching xrefs 1079, 1081
 content area 44
 creating tool palettes 96
 docking 45–46
 preventing 46
 inserting layouts 237–238
 overlaid xrefs 1081
 resizing 45
 roll-over behavior 45
 tree view. *See* tree view (DesignCenter)
 undocking 45
 See also DesignCenter
destination applications. *See* container applications
detaching 1087
 xrefs 1087
Developer Help 4
 starting 4
dialog boxes 10, 35, 38–39
 Help on 10
 suppressing when entering commands 38
switching between the command line and 38
system variables affecting 39
transparently opened, change delays 35
diameter dimensions 876, 891, 915, 918, 932
 creating 915, 918
 definition points 932
 placing dimension text for 891
dictionaries for spelling checker 848–850
 custom 848–850
 creating 850
 editing 848–850
 switching 849

diffuse color (of materials)
 See also diffuse reflection

diffuse reflection
 See also diffuse color (of materials)

digital signatures
 See also signed files

digitizing 335

 entering 3D Cartesian coordinates by 335

DIMASO system variable 879

DIMASSOC system variable 878–879

DIMCEN system variable 916

DIMCENTER command 916

DIMDIAMETER command 915

dimension geometry 877, 883

 controlling 883

dimension line arcs (for angular dimensions) 920

dimension lines 877, 883–884, 888–892, 895–896, 920, 930

 aligning dimension text with 892, 895

 arcs 920

 control options 883

 forcing an internal line 890–891

 modifying 884

 placing dimension text 889, 891, 896

 above 896

 within/outside of 889, 891

 properties 883

 spacing for baseline/continued dimensions 930

 trimming of 888

dimension scale 906–908

 setting 907–908

 for dimensioning in layouts 908

 the overall dimension scale 907

 for scaling model space

 dimensions to layouts 908

 values affected/not affected by 906

dimension style overrides 881, 926–927

 applying 927

 creating 926–927

listing 881

dimension styles 880–882, 925

 applying 925

 comparing 881–882

 creating 881

 listing 881–882

 listing settings for 881–882

 overrides. *See* dimension style overrides

restoring 925

standards. *See* standards (for named objects)

See also named objects

dimension text 794, 877, 889, 891–898, 928–930

 aligning 892, 895, 930

 with dimension lines 892, 895

 controlling 889

 home position 928–929

 returning text to 929

 modifying 928

 moving 928

 placing 889, 891, 893–896

 above dimension lines 896

 for diameter dimensions 891

 horizontally 893, 895

 manually 893–894, 896

 at the second extension

 line 895

 vertically 894, 896

 within/outside of dimension lines 889, 891

 prefixes and suffixes 897

 replacing 928, 930

 rotating 928–929

 text styles 897–898

 user text 897

 values. *See* dimension values

See also leader text

dimension text paragraph break symbol 897

dimension text separator symbol 897

dimension values 899–903

 rounding off 901–902

 units 899–900

 adding and formatting 900

zero suppression 902–903

dimension variables. *See* dimensioning system variables

dimensional input 359, 363
 changing settings 363
 lock icon 359
 tooltips, specifying points in 359

dimensioning 876, 906
 basic concepts 876
 methods 906
 types 876
 See also dimensions

dimensioning system variables 881
 listing 881

dimensions 80, 82, 205, 325, 664, 876–878, 903, 905, 908, 925, 932, 934–935
 angular. *See* angular dimensions
 associating/reassociating 934–935
 association points 934
 associative. *See* associative dimensions
 associativity. *See* associativity (of dimensions)
 basic dimensions 903, 905
 in the Block Editor. *See* parameters (in dynamic blocks)
 creating
 See also dimensioning
 creating tools for 80
 definition points 932
 draw order 325
 changing 325
 elements 877
 exploded dimensions. *See* exploded dimensions
 exploding 664
 filtering associative/nonassociative selections 878
 linear. *See* linear dimensions
 modifying (editing) 925
 nonassociative. *See* nonassociative dimensions
 radial. *See* radial dimensions
 restyling 925
 scaling
 See also dimension scale

styles. *See* dimension styles

text. *See* dimension text

types 876
 using a tool created from 82

values. *See* dimension values
 See also dimensioning

DIMGAP system variable 821

DIMLINEAR command 843

DIMRADIUS command 915

DIMREASSOCIATE command 878–879, 934

DIMREGEN command 879

DIMSCALE system variable 907

direct distance entry 328, 387
 drawing lines 387
 specifying points 387

direction 344, 696, 861
 extruding objects by 696
 of axes, positive, determining 344
 of table text 861

direction grips 755

disassociating 935

compound objects. *See* exploding compound objects

dimensions 935

discarding 1102
 changes for in-place reference editing 1102

display (screen). *See* screen display

Display Alert upon Standards Violation option (CAD Standards Settings/Check Standards dialog boxes) 148

display drivers

Display Lineweight dialog box 321

display performance 323

Display Plot Area option (Page Setup dialog box) 954, 980

Display Standards Status Bar Icon option (CAD Standards Settings/Check Standards dialog boxes) 148

display. *See* screen

displayable attributes. *See* labels

displaying 11, 40, 155, 325, 713, 732, 736, 762

cameras

command line when hidden 40
cutaway geometry 762
draw order and object display 325
grip tools 736
history of composite solids 732
interferences 713
original solids from composite
solids 732
Quick Help (Info palette) 11
selected layers 155
shadows
distance and angle coordinate
display 329
distance multiplier action override
property 516–517
specifying 516–517
Distance Type property 499–500, 504,
506
for move actions 499–500
for stretch actions 504, 506
distances 378–379, 387, 391–392, 408,
665, 847
3D view
calculating 391–392
converting between model space and
paper space 847
entering directly. *See* direct distance
entry
fuzz distance 665
measuring the distance between two
points 408
polar distances 378–379
specifying 387
distant lights 1118, 1120–1121, 1123,
1126
adding 1118, 1120–1121
source vectors 1123, 1126
dithering 1009, 1011
turning off 1009, 1011
dividing 388, 390–391
objects 388, 390–391
into equal segments 388, 390–
391
dockable windows, in workspaces 69
docking 24, 40–41, 45–46
command window 40–41
DesignCenter 45
preventing 46
toolbars 24
docking windows 64, 88
doglegs (for dimension text) 915
doglegs (hook lines) 817
dollar signs (\$) 868, 1106
bound named object name
dividers 1106
formula cell address fix value
prefix 868
donuts 322–323, 442, 664, 783, 788
creating 783
drawing 442
exploding 664
line widths, changing 322
modifying 788
simplified display 323
dotted grid 375
double-byte characters 128
renaming drawings named with 128
double-click editing 274, 611
DOWN ARROW key, viewing/selecting
options in dynamic
prompts 360
down-arrow icon 361
in dynamic input tooltips 361
downloading online content 55, 58
dragging 162, 615, 641, 770, 773
hatch patterns into drawings 770,
773
lengthening objects by 641
panning by 162
rotating objects by 615
zooming by 162
dragline. *See* rubber-band line
draw order 325, 772, 789
of hatches 772, 789
modifying 789
of dimensions 325
changing 325
of objects 325
changing 325
of text 325
changing 325

drawing 196, 204, 217, 318, 376, 379, 387, 417–422, 424–425, 427, 430, 432, 434–446, 448–451, 469, 682–693
3D solid primitives
arcs 432, 434–436
specifying three points 435
tangent to arcs 436
tangent to lines/arcs 434
using start point, center point, and endpoint 435
boxes 682
circles 436–437
tangent to objects 436–437
cones 684–686
construction lines 450–451
cylinders 686–687
donuts 442
ellipses 443–444
elliptical arcs 445
equilateral triangles 424–425
geometry 469
for dynamic blocks 469
helixes and spirals 448
isometric circles 443–444
lines 196, 318, 376, 387, 418–419, 434–435
in viewports 196
in model space 318
tangent to arcs 434–435
using direct distance entry 387
using orthogonal locking 376
in model space 204
multilines 427
objects 217, 379, 417, 469
for dynamic blocks 469
in layout viewports 217
using polar distances 379
using polar tracking 379
polygons 420, 424–425, 438
polylines 318, 419, 421, 430, 438, 440
freehand 430
in model space 318
polysolids 692–693
pyramids 689–690
rays 451
rectangles 424–425
reference points 449
spheres 688–689
splines 446
squares 424–425
tori 691
wedges 683
wide polylines 420, 422, 439, 441
See also drawings
drawing (DWG) files 119, 122, 127, 129–131, 134–135, 243, 249, 464, 466, 474, 557, 559, 574–575, 844
as blocks 122, 464, 466, 474, 557, 559, 574–575
creating 464, 466
inserting 122, 557, 559
opening 474
updating block definitions from 574–575
auditing 134
checking 122
for authenticity 122
file name extension 127
finding 119, 130–131, 844
for sheet sets/views 243, 249
consolidating 243
inserting as blocks 122
repairing damaged files 134–135
saving block definitions as 130
saving selected objects into 127, 129
transmitting on the Internet. *See* transmittal sets
See also drawings
drawing area (in application)
window 27, 62, 64, 66
clearing the screen 62
coloring elements in 62, 66
font selection (for windows) 64
right-clicking in 27
turning off shortcut menus in 27
drawing boundary. *See* drawing limits
drawing content. *See* content
(DesignCenter)

drawing display
 changes stored in profile/saved to workspace
drawing extents 161, 163
 zooming to 161, 163
drawing interchange binary files. *See* DXB
 (drawing interchange binary) files
drawing interchange files. *See* DXF
 (drawing interchange format) files
drawing limits. *See* grid limits
drawing orientation 964, 985
 setting 964
drawing planes (in 3D space) 343, 349
 specifying 343
drawing projects. *See* projects
drawing properties 119–120, 130
 changing 119
 creating additional 119
 defining 120
 displaying 119
 in fields (in text) 119
 finding drawing files with certain properties 119, 130
Drawing Properties dialog box 120
 defining drawing properties 120
Drawing Recovery Manager 138–139
 expanding/collapsing nodes in 139
 opening 138–139
 removing drawings from 139
drawing sets 240, 1031, 1038, 1041, 1049
 creating 1031
 including model layouts in 1031, 1038, 1041
 publishing 1031
 republishing 1049
drawing sheets (in drawing sets) 1031–1039, 1041–1042
 adding 1031, 1033, 1038, 1041
 copying 1032, 1035, 1039, 1042
 page setups for 1032, 1036–1037, 1039, 1042
 changing 1032, 1036–1037, 1039, 1042
 importing 1037
 removing 1032, 1034, 1039, 1041
renaming 1032, 1035, 1039, 1042
reordering in the sheet list 1032, 1034, 1039, 1041
drawing standards. *See* standards (for named objects)
drawing templates/template files 74, 108–112, 214, 236, 243
contents common to 111
creating 111–112
for creating sheets 243
default file. *See* default drawing template file
default location 111
for extracting attribute information.
 See attribute extraction template files
file name extension 111
importing layouts from 214
starting drawings with 74, 108–112
 See also templates (template files)
drawing tools (precision tools) 328
drawing units 114–118, 205, 207, 328
angular unit conventions 118
converting 114–115, 117
coordinate unit types 328
determining 114, 205
linear unit conventions 115
precision 116
rounding off of 116
setting 114, 116, 205, 207
 See also architectural units
 See also imperial units
 See also metric units
DRAWINGRECOVERY command 138
drawings 47, 50–51, 53, 59, 62, 74, 96, 107–112, 114, 122–124, 126–130, 135–137, 142, 144–146, 149, 152, 166, 180, 204–205, 210, 249, 253–254, 288, 461, 465, 610–611, 770, 957, 964, 985–987, 996–997, 1017–1018, 1078–1081, 1087–1089, 1091–1093, 1106, 1110–1112
accessing 47
adding from/to tool palettes 51
adding hatch patterns to 770

associating standards files with 144
attaching digital signatures to. *See*
 files, attaching digital
 signatures to
attaching xrefs to 1078–1081, 1088,
 1093
 with DesignCenter 1079, 1081
 overlaying xrefs on vs. 1088
background colors 62
binding 1091–1092, 1106
 dependent named objects
 to 1106
 xrefs to 1091–1092
checking 142, 146, 149, 152
 for standards violations 142,
 146, 149, 152
comments on. *See* markups
content. *See* content (for drawings)
 (DesignCenter)
converting to a different plot style
 type 996–997
copying 288, 465
 layers between 288
 paper space objects
 between 465
copying between 126
correcting. *See* markups
creating 96
 tool palettes from 96
creating drawing template files
 from 111–112
default folder 122–123
 changing 122–123
design environment. *See* model space
detaching xrefs from 1087
dimensioning. *See* dimensioning
displaying 126, 166
 in the Aerial View window 166
 on the Windows taskbar 126
downloading content to 59
exporting
 visual styles 180
 See also plotting to files
files. *See* drawing (DWG) files
identifying information in. *See*
 drawing properties

importing 180
 visual styles 180
layout environment. *See* paper space
marking up. *See* markups
multiple open drawings 126
multiple. *See* multiple drawings
naming 128
 for international work 128
offsetting (on paper) 985
opening 50, 53, 122–124, 249, 253–
 254
 authenticity check 122
 with DesignCenter 50, 53
 in Windows Explorer 122
 partially 123–124
 within sheet sets 249, 253–254
orientation 964, 985
overlaying xrefs on 1089
panning. *See* panning
partially open. *See* partially open
 drawings
pasting objects into 610–611
plotting. *See also* plotting
preparing for plotting (process
 steps) 210
previewing 130, 1017–1018
 for plotting 1017–1018
printing. *See* printing
properties. *See* drawing properties
reducing 461
 file size 461
removing standards files from 145
renaming drawings named with
 double-byte/special
 characters 128
repairing open drawings 135
restoring from backup files 136–137
saving 127–130, 136, 1110–1112
 automatic saves 129, 136
 incremental saves 127
 with layer indexes 1110–1112
 partial saves 127, 129–130
 with preview images 130
 reducing the time required
 for 127

saving backup files when 129, 136
with spatial indexes 1110–1112
to different formats 127
scaling 114, 204–205, 957, 985–987
to fit the paper 957, 986–987
for plotting 205, 957, 985
signing. *See* digital signatures
standards for. *See* standards (for
named objects)
starting 74, 107–112
 with the default drawing template
 file 109–110
 with drawing templates/template
 files 108–112
 with templates 74
 with wizards 111
switching between 126
templates. *See* templates (template
files)
titles. *See* title blocks
with xrefs, archiving 1091
zooming. *See* zooming
DRAWORDER command 325
DS\$ (sheet set data backup) files 244
DSD (Drawing Set Descriptions)
 files 1028, 1033, 1050
 adding drawing sheets to drawing sets
 from 1033
 saving publish options to 1050
DST (sheet set data) files 242, 266
 team member access to 266
 update of data from team member
 changes 266
duration of animation 190
DWF (Design Web Format) files 261,
 265, 1019, 1027–1028, 1040,
 1042–1043, 1045, 1050–1051,
 1053, 1068–1073
compression 1070
 specifying 1070
distribution options 1028
font-handling 1071–1072
 editing the fonts eligible for
 capture list 1072
 specifying 1071
format 1019
layers displayed in 1019
minimizing file size 1071–1072
named views displayed in 1019
opening/viewing/plotting 1019
output directory 1051
 changing the default 1051
output type 1051
 setting 1051
panning/zooming in 1019
password security 1053
 specifying 1053
pen settings 1072–1073
 editing 1072–1073
plot files. *See* DWF plot files (plotted
 DWF files)
plotting 1019, 1028
published 265
 sheet properties in 265
published files. *See* published DWF
 files
publishing 261, 1027
 for distribution 1027
 property options for 261
publishing electronic drawing sets
 to 1028, 1040, 1042
publishing sheet sets to 1043, 1045
publishing sheet subsets to 1045
resolution 1068–1069
 specifying 1068–1069
viewers 1019
viewing 1027–1028, 1050
 in Autodesk DWF Viewer 1027,
 1050
DWF plot files (plotted DWF files) 1051,
 1065–1067
background color 1065
output directory 1051
 changing the default 1051
settings 1065–1067
 color depth 1067
 specifying 1066
DWF6 ePlot.pc3 plotter configuration
 files 1040, 1065–1072
creating 1065–1066
recreating the default file 1065

settings 1065–1072
color depth 1067
file compression 1070
font handling 1071–1072
resolution 1068–1069
specifying 1066

DWF6 files 1028, 1064
plotting 1064
See also DWF (Design Web Format) files

DWF6 plotter drivers 1064
configuring 1064
See also DWF6 ePlot.pc3 plotter configuration files

DWG (drawing) files
underlays. *See* DWG underlays

DWG file name extension 127, 236

DWG files. *See* DWG (drawing) files

DWG to PDF driver 1022

DWG TrueConvert (Migration Tool) 2

DWGCHECK system variable 122

DWS (standards) files
See also standards files

DWT (template) files 266
See also drawing templates/template files
team member access to 266

DWT file name extension 111, 236

DXB (drawing interchange binary) files 1020
creating (plotting) 1020
format 1020

DXB driver 1020

DXF (drawing interchange format) files 571
format 571

Dyn button (status bar) 359

dynamic block definitions 468, 470, 482, 484, 487–496, 498–500, 502–504, 506–508, 510–516, 518, 520, 523, 531, 549–551
adding actions to 468, 470, 482, 496, 498–500, 502–504, 506–508, 510–516, 531
array actions 513–514
flip actions 511–512

lookup actions 515–516, 531
methods 470
move actions 498–499
overview 468, 482, 496
polar stretch actions 506–507
rotate actions 508, 510
scale actions 500, 502
stretch actions 503–504
adding parameter sets to 518
adding parameters to 468, 470, 484, 487–496, 518, 520, 523
alignment parameters 492
base parameters 496
flip parameters 493
grips and 520
linear parameters 488
lookup parameters 495
methods 470, 518
overview 468, 484
point parameters 487
polar parameters 489
rotation parameters 491
visibility parameters 494, 523
XY parameters 490

correcting 550–551
deleting elements from 550
modifying 549
opening 549

dynamic blocks 460, 467–471, 474, 479, 481–482, 484, 531, 535, 537, 541, 550–551, 560, 562
3D solids in 471
actions. *See* actions (for dynamic blocks)
adding elements to. *See* dynamic block definitions
assigning data to 531
base points 471
behavior 460
changing data associated with 531
creating 467–468, 482, 484
dependencies (element relationships) in 470, 551
dependency highlighting 479, 481
designing 469
drawing geometry for 469

dynamic block references 560, 562
editing by changing lookup properties 537
exploding 541
grips. *See* grips (for parameters) (custom grips)
lookup tables 535
manipulating 562
opening 474
overview 468
parameters. *See* parameters (for dynamic blocks)
properties. *See* action override properties
properties. *See* parameter properties
resetting after manipulation 562
scaling 541
testing 471, 550
dynamic coordinate display 329
Dynamic Input 358, 363
 turning on/off 358, 363
 using 358
dynamic input tooltips 361
 correcting typing errors 361
 entering coordinate values in 361
dynamic panning (Realtime option) 160
dynamic prompts (tooltips) 360, 362, 364
 choosing options 362
 displaying 364
dynamic updating of the Aerial View window 166–167
 turning on/off 167
dynamic viewing (of 3D objects)
dynamic zooming (Realtime option) 161

E

e-Learning Catalog option, Subscription Center 13
e-learning lessons (Autodesk) 12
earth azimuth 63
 display options 63
earth horizon 63
 display options 63
edge-defined meshes 716, 721

edges 632, 716, 721
 boundary edges 632
 cutting edges 632
 mesh edges 716, 721
 Coons surface path
 meshes 716, 721
edges (of 3D solids)
 customizing display
EDGESURF command 716, 721
Edit shortcut menu 27
 displaying 27
editing 36, 219, 259, 279, 301, 413, 564, 566, 603, 611, 655, 670, 672, 674, 676, 733, 744, 772, 794, 824, 843–844, 848–850, 857, 952, 960, 983, 1001, 1008, 1057, 1072, 1095–1096, 1098
3D solid edges
3D solid faces
3D solid vertices
attribute definitions 564, 566
block references, in place 1096, 1098
block template (BLK) files 1057
calculator variables 413
in the command window 36
composite solids 733
custom paper sizes 952, 983
dictionaries (custom) 848–850
double-click editing 611
fields (in text) 824
fonts eligible for capture list 1072
groups 603
hatch boundaries 772
helixes and spirals 672
layer 301
 settings 301
layers, controlling 279
in layout viewports 219
multiline 674, 794
 styles 674
 text 794
multiline styles 674, 676
multiline text 843–844
objects 611, 655
 with grips 655

methods 611
plot style tables 960, 1001
plot styles 1001, 1008
separating 3D solids into original shapes 744
sheet/sheet set properties 259
splines 670, 672
 with grips 670
tables 857
text. *See* editing text
xrefs 1095–1096, 1098
 in place 1096, 1098
 in a separate window 1096
See also modifying
editing commands 597, 675
 commands not for multilines 675
 entering before selecting objects 597
 selecting objects before entering 597
editing text 816, 842, 847, 864, 867
 leader text 847
 single-line text 842
 stacked text 816
 in table cells 864, 867
educational products 1080
electronic drawing sets 1027–1028,
 1040–1042, 1050
 creating 1041
 plotting published sets 1050
 publishing 1027–1028, 1040, 1042
 for distribution 1027
 to DWF files 1028, 1040, 1042
 viewing published sets 1050
electronic markup feature. *See* markups
 (markup objects)
electronic sheet sets 1043
 publishing 1043
elevation (default Z value) 349
 in UCSs 349
ELLIPSE command 359
 dimensional input 359
ellipses 393, 443–444
 calculating areas/perimeters 393
 drawing 443–444
See also objects
elliptical arcs 189, 445
 drawing 445
motion path animations and 189
elliptical-based cones 685
elliptical-based cylinders 687
ENTER key 34, 865
 executing commands 34
 moving down one cell 865
 repeating commands 34
entering 34, 36, 38, 328, 331–332, 345,
 357, 361, 597, 864–865, 868, 871
 commands 34, 38
 displaying prompts on the
 command line 38
 on the command line 34
 suppressing dialog boxes 38
coordinate values 361
 in dynamic input tooltips 361
coordinates 328, 331–332, 345, 357
 broken pencil icon warning 357
 relative to the WCS 345
 relative values 328, 331–332
editing commands 597
 after selecting objects 597
 before selecting objects 597
formulas 868, 871
 in table cells 868, 871
system variables, on the command
 line 36
text 864–865
 in table cells 864–865
environment 72
 customizing, with profiles 72
environment maps. *See* reflection maps
environment profiles. *See* profiles (of
 environment settings)
environment settings 76
 precedence 76
environment variables 76
 precedence of settings 76
EPS (Encapsulated PostScript) files 1024–
 1025
 format 1024
 plotting 1024–1025
equal sign (=) 868, 871
 formula start character 868, 871
equilateral triangles 424–425
 drawing 424–425

erased objects 607–608
restoring 607–608
erasing 430–431, 605
freehand lines 430–431
undoing 605
erasing objects 607–608
error reporting (to Autodesk) 13, 138–139
 Resolved Error Reports 13
 turning on/off 139
errors 361, 550–551, 1104
 correcting in dynamic block definitions 550–551
 correcting mistakes. *See* undoing actions
 in dynamic input tooltips, correcting 361
 messages, from xrefs 1104
ESC key 35, 606
 canceling commands 35, 606
evaluating mathematical expressions 403, 414
 in dialog boxes 403, 414
executing 34
 commands 34
external references (xrefs) 85
 placing from tool palette 85
 prompt for rotation angle 85
expanding text 795
EXPLODE command 423
exploded dimensions 878, 933
 modifying 933
exploding 541, 583–584, 664–665
 compound objects 583–584, 664–665
 associative hatches 664
 blocks 583–584, 665
 donuts 664
 polylines 664
 xrefs 665
 dynamic blocks 541
 objects 665
EXPORT command 465
export file formats 571, 610
 CDF files 571
 DXF files 571
SDF files 571
WMF format 610
See also plot file formats
exporting 72, 102, 153, 180, 191, 300, 860, 1022, 1062–1063
 animation files
 animations 191
 batch audit reports 153
 block attributes. *See* extracting attribute information
 drawings 1022, 1062
 as 3D DWF files 1062
 as PDF files 1022
 environment profiles 72
 layer settings 300
 objects 1063
 to 3D DWF files 1063
 tables 860
 tool palettes 102
 visual styles 180
See also export file formats
expressions (in QuickCalc) 400, 404, 406–407
calculating (evaluating) 404
changing the font color in the History area 406
copying from the History area 406–407
entering and retrieving 400
precedence rules 404
syntax 404
EXTEND command 634–635
extending 632, 634–636, 643, 649
 chamfered objects 649
 filleted objects 643
 objects 632, 634–636
 in 3D 635–636
 while trimming 632
 spline-fit polylines 635
 wide polylines 634
extending objects
 See also stretching
extension line origins 909, 934
 as association points 934

extension lines 877, 884, 886, 895, 909, 915, 923, 934
for arc length dimensions 923
control options 884
fixed-length lines 886
modifying 886
oblique 915
origins of 909, 934
placing dimension text at the second 895
properties 884
extension origin offset 931
changing 931
Extents Plot Area option (Page Setup dialog box) 954, 980
extents. *See* drawing extents
external databases
See also database objects
external references (xrefs) 80, 85, 249, 465, 557, 665, 828, 1077–1081, 1083, 1087–1093, 1095–1096, 1098, 1101–1102, 1104–1105, 1107, 1109–1112
attaching 1079–1081, 1088
with DesignCenter 1079, 1081
overlaying vs. 1088
tools for 1079
binding 1091–1092
to drawings 1091–1092
vs. blocks 465
bound xref name prefix 1101
changed xrefs 1090
accessing 1090
notification of 1090
circular references 1105
clipped 1093
clipping 1095, 1110
contextual fields in 828
copying, for demand loading 1111–1112
creating tools for 80
defined 1077
detaching 1087
discarding changes to 1102
drawing files for sheet views as 249
in drawing files inserted as blocks 557
drawings with, archiving 1091
editing 1095–1096, 1098
in place 1096, 1098
in a separate window 1096
error messages 1104
exploding 665
External References palette 1081
freezing layers of 1110
from Autodesk Educational Product 1080
layer properties 1079
listing 1107
log files, turning on/off 1109
nested. *See also* nested xrefs
notification of attached xrefs 1079, 1083
relocated xrefs 1083
notification of changed xrefs 1090
OLE objects in 1102
overlaying 1081, 1088–1089
vs. attaching 1088
with DesignCenter 1081
reloading 1090
saving back changes to 1101–1102
saving with indexes 1110–1112
unloading 1109–1110
updating 1078, 1090–1091
with demand loading turned on 1090
uses 1078
using a tool created from 85
viewing by type 1081
External References palette 1081
viewing referenced files by type 1081
extracting 484, 539, 564, 566–568, 573, 723
attribute information 564, 568, 573
block attribute data 566
block attribute information 567
custom properties (in dynamic blocks) 539
edges for wireframes 723
parameter values 484

EXTRUDE command 696
 See also SWEEP command
extruding 728
 objects 728
 to create solids 728
 to create surfaces 728

F

F1 key, Help resource 10
F12 key, for Dynamic Input 359, 363
F2 key, switching from the command window to the text window 37
F3 key 365
 turning running object snaps on/off 365
faces
 3D faces. *See* faces (of 3D solids)
faces (of 3D solids) 752
 back faces. *See* back faces
 colors
 edges. *See* edges (of 3D solids)
 face styles
 sectioning with 752
 transparency
facet edges
faceted faces
faceted surfaces. *See* meshes
faceted surfaces. *See* surface meshes
FACETRES system variable 681
fade (fading) 1097
 of objects not in working sets 1097
falloff angle (of spotlights) 1124
 setting 1124
falloff of light. *See* attenuation of light
Favorites folder (DesignCenter) 47–49
 adding items to 47–48
 displaying the contents 49
feature control frames 936, 938
 compartments 936, 938
features (of objects), material condition symbols 936, 938
feet 116, 404, 902–903
 QuickCalc formats 404
 specifying 116
 suppressing zero values 902–903

field angle (spotlights) 1124
fields (in formulas) 872
 inserting 872
 in table cells 872
fields (in text) 119, 795, 798, 823–826, 828–830, 868
 adding hyperlink fields 830
background 824
compatibility with AutoCAD LT and previous releases of AutoCAD 829
contextual fields 828
displaying object properties in 825
drawing properties in 119
editing 824
formatting options 824
inserting 823–826, 868
 as placeholders 823, 826
 in table cells 825, 868
invalid field display 823
no value in field display 823
preserving values in 824
selecting 824
text style 824
updating automatically 829
updating manually 829
 updating multiple fields 829
fields. *See* database record fields
file compression 1070
 of DWF files 1070
 specifying 1070
file formats 570–571, 610, 1019–1020, 1022, 1024–1025
 attribute extraction files 570
 CDF files 571
 DWF files 1019
 DXB files 1020
 DXF files 571
 HP-GL format 1025
 HP-GL/2 format 1025
 PDF files 1022
 PostScript files 1024
 SDF files 571
 WMF files 610
file management. *See* Autodesk Vault
FILEDIA system variable 39

files 567–568, 1019–1020, 1022, 1024–1026
attribute template files 568
block attributes in 567
plotting to 1019–1020, 1022, 1024–1026
DWF files 1019
DXB files 1020
plot files 1026
PostScript files 1024–1025
raster files 1022
signing. *See* digital signatures
support. *See* support files
temporary. *See* temporary files
Files tab (Options dialog box) 131
Fill mode, turning on or off 323
fill patterns, scaling 1015
fill styles 1016–1017
assigning 1016–1017
FILLET command 642
fillet radius 643, 645
overriding 645
setting 643, 645
filleted objects 643
trimming and extending 643
filleting (rounding) 642–647
3D solid edges
3D solids
creating fillets 642
hatch boundaries 642
line segments 646
lines with polylines 644
multiple sets of objects 642, 647
objects 642, 645–646
in 3D 645
without trimming 646
parallel lines 645
polylines 644, 646
setting the radius 643, 645
specifying locations for 643
fillets (rounds)
creating. *See* filleting objects
fills 798, 862, 1015–1017
assigning styles 1016–1017
multiline text object fills 798
scaling patterns 1015
table colors 862
filtered lists, saving 596
filtering 152, 280–281, 288, 296–298, 594, 597
batch audit report data 152
layer names 280–281, 288
layer names by selecting layers 297
layers, by layer property 296
layers, by name 296
nested layer filters 298
object selection 597
selection sets 594
filters 280–281, 294, 594, 596
layer filters 280–281, 294
named object selection filters 594, 596
saving 281
See also coordinate filters
FIND command 844
Find tool 119
finding drawing files 119
finding 57–58, 119, 130–131, 844–845
drawing files 119, 130–131, 844
online content 57–58
text 844–845
multiline text 844–845
fit points (for spline curves) 670
fixed-length extension lines 886
flat lighting
FLATSHOT command 764
flattened views of 3D objects 764
flip actions 511–512
adding to dynamic block
definitions 511–512
parameter supported by 511
flip grips 520, 561
flip parameters 482, 484, 493, 518
actions supported by 482, 484
adding to dynamic block
definitions 493
grip type 482
sets 518
Flip Set parameter set 518
flipped arrowheads 887
appearance in earlier versions 887

flipping 888
 arrowheads 888

flipping. *See* mirroring

floating properties, block insert
 settings 553

floating. *See* undocking

flying through models
 settings
 simulating

FMP (font mapping) files 837

folders 48, 96, 122–123, 131–132
 creating tool palettes from 96
 default drawing folder 122–123
 changing 122–123
 DesignCenter Home button folder,
 changing 48
 My Documents folder 122, 131–132
 specifying the search path 131–132
 See also Favorites folder

Folders tab (DesignCenter) 47

font mapping files 836–838
 default table 838
 for PostScript fonts 836
 specifying 838

FONTALT system variable 836

fonts 802, 804, 833–838, 852, 1071–1072
 alternative font 836, 838, 852
 format code 852
 assigning to text styles 833, 836
 Big Fonts 834–835
 changing multiline text fonts 804
 converting 837
 DWF file font-handling 1071–1072
 editing the fonts eligible for
 capture list 1072
 specifying 1071
 modifying, with text styles 833
 proxy fonts 833, 837
 smoothing 802
 specifying 835
 styling 833
 substitute fonts 836

foreshortened radius dimensions. *See*
 jogged radius dimensions

format codes, for multiline text 852

formatting text 795, 798–799, 803–804,
 806, 814, 824, 832, 851, 861–865
 character formatting 832
 in fields 824
 in table cells 861–865
 multiline text 798–799, 803–804,
 806, 851
 in alternate text editors 851
 as lists 806
 stacked text 814
 See also text styles

formulas 868–872
 cell address fix value prefix 868
 copying to other table cells 868
 entering in table cells 868, 871
 fields in 872
 inserting in table cells 872
 inserting in table cells 868–870
 start character 868, 871

fps (frame rates) 190–191
 animation 190–191

fractions 814–815, 817
 stacking 814–815
 unstacking 817

frame rates 190–191
 animation 190–191

free orbit in 3D Orbit view

freehand lines (sketching) 430–431
 accuracy on slow computers 430
 erasing 430–431
 linetypes 430
 saving 431
 in Tablet mode 430

freezing 194, 225–228, 278, 283, 1110

layers 194, 225–228, 278, 283, 1110
 in layout viewports 194, 225–228
 in paper space 228
 of xrefs 1110

From option (object snaps) 388

front, moving objects to 325
 draw order of objects 325

frozen layers 225–226, 229, 279, 283–284
 creating 226, 229

functions 411
calculator variables 411
fuzz distance 665

G

gap tolerance in hatch boundaries 770, 775
Generate Section/Elevation dialog
box 764
geographic locations of models 1127–1128
geometric 903, 936–937, 939
characteristics 936
points. *See* points (geometric)
tolerances 903, 936–937, 939
composite tolerances 939
creating 936–937
creating with leaders 937
projected tolerances 939
geometric characteristic symbols 936
geometric objects 80, 82
drawing, creating tools for 80
using a tool created from 82
geometry 125, 469, 523, 528–530, 877
dimension geometry 877
for dynamic blocks
drawing 469
making objects invisible in
visibility states 529
making objects visible in visibility
states 528–529
See also action selection sets
showing/hiding 523, 530
loading into partially open
drawings 125
getvar AutoLISP function 404
GIS blocks, accessing collections of 58
global constants 411, 413–414
accessing 411
accessing from dialog box or
window 414
creating 411
specifying from the QuickCalc Input
box 413

Global Scale Factor value (linetype
scale) 315
glyphs 188
camera glyphs 188
Gooch face style
gradient fills 80, 323, 783–785, 789
creating 784–785
creating tools for 80
modifying 785, 789
simplified display 323
graphical objects. *See* objects
graphics area. *See* drawing area
graphics cards
graphics cursor. *See* crosshairs
graphics window. *See* application window
grayscale 1010–1011
converting colors to 1010–1011
grid (3D Orbit view)
grid (drawing area) 371–375
color of 371
displaying 373–374
dots and lines 375
overview 371
rotating 375
spacing 372–375
setting 372, 374–375
vs. snap grid spacing 373
turning off display 372
See also snap (snap grid)
grid limits 161, 163, 206, 371, 374
calculating 206
overriding 371
setting 374
zooming to 161, 163
grid snap. *See* snap
gridlines (tables) 863
grip modes 655–656
cycling through 655
grip tips, displaying 657
grip tools 735, 737–738, 741
default location 737
in 3D views 735
moving 737
moving objects with 738
rotating objects with 741
types of 735, 737

GRIPDYNCOLOR system variable 561
 changing color of custom grips 561
GRIPOBJLIMIT system variable 656
grips 186, 360, 470, 654–661, 663, 670,
 728, 754, 842–843, 856–858, 928
 3D solid subobjects
 3D solids and 728
 base grips 655
 in blocks 663
 camera grips 186
 canceling grip selection 657
 copying objects 660–661
 defined 655
 dynamic blocks and 470
 editing 655, 670
 objects 655
 splines 670
 grip tools in 3D view. *See* grip tools
 hot grips 655
 hover color 657
 limiting the display of 656–657
 mirroring objects 656, 659, 663
 retaining the originals 663
 moving 656, 658, 928
 dimension text 928
 objects 656, 658
 moving multiline text 843
 multiple hot grip selection 655
 for parameters. *See* grips (for
 parameters) (custom grips)
 quadrant grips 655
 rotating objects 656, 658
 scaling objects 656, 659
 section objects 754
 selecting 655
 stretching objects 656–658
 surfaces and 728
 table cells 856–858
 for text objects 842
 tips 657
 turning on/off 657
 using with dimensional input 360
grips (for parameters) (custom
 grips) 479–482, 484–485, 520–
 523
 as displayed in the Block Editor 482
dependency highlighting 479, 481
displaying in Block Editor 479
insertion cycling 521, 523
modifying order 523
overview 484
repositioning 520, 522
resetting 520, 522
specifying 485, 520, 522
specifying color 480
specifying size 480
types 482, 520
ground horizon 63
 display options 63
ground origin 63
 display options 63
ground plane 63
 display options 63
ground plane grid 63
 display options 63
group selection, toggling 602–603
groups (of objects) 601–604
 adding objects to 603
 vs. blocks 601
 creating 602
 cycling through objects in 602
 editing 603
 removing (ungrouping) 604
 removing objects from 603
 reordering objects in 603–604
 selecting 602
 toggling group selection 602–603
GTAUTO system variable 737
GTDEFAULT system variable 737
guide curves 700, 729–730
 for lofting 729–730

H

hanging indents, creating 812
hardware 309
 acceleration
 linetype use 309
hardware acceleration
hatch angle 789
 changing 789

hatch areas 774–775, 777, 781, 788
hatching 774–775, 781
 with a solid pattern 781
holes within 788
islands within 775, 777, 788
 creating 788
 removing 777
planar requirement 775
hatch associativity 772, 788
 loss of 772, 788
 removing 772, 788
 restoring 788
hatch boundaries 642, 647, 770, 772,
 774–775, 778–780, 788
 bridging gaps in 770, 775
 chamfering 647
 in complex drawings. *See* boundary
 sets
 defining 770, 774–775, 778–780,
 788
 with boundary points 779–780
 as boundary sets 778
 with internal points 774
 redefining 788
 editing 772
 enclosed areas within. *See* islands (in
 hatch areas)
 filleting 642
 gap tolerance 770, 775
 hiding 779
 modifying 788
 recreating 788
 See also boundary sets
HATCH command 770, 772, 775, 778
hatch objects 367
 setting object snaps to ignore 367
hatch patterns 52, 205, 770–773, 781,
 785–789
 adding to drawings 770
 adding to the program supply 771
 angle 789
 changing 789
 behavior 771
 choosing 771, 785
 default file 773
 defining custom patterns 773
density limit 772, 787, 789
 setting 772, 787, 789
dragging into drawings 770, 773
enclosed areas within. *See* islands (in
 hatch areas)
ISO patterns 771, 785
loading the DesignCenter content area
 with 52
location 771
modifying 788
predefined patterns 771, 785–786
 using 786
scaling 205
solid fills 781
solid, *See also* gradient fills
user-defined patterns 786
hatches 50, 80, 84, 323, 771–772, 779,
 781, 787–789
 adding to tool palettes 50
associative hatches 772, 788
associative. *See* associative hatches
associative. *See* associative hatches
associativity 772, 788
boundaries. *See* hatch boundaries
creating, creating tools for 80
creating. *See* hatching
draw order 772, 789
 modifying 789
islands in. *See* islands
modifying 787
nonassociative hatches 772, 788
origin point 771
patterns. *See* hatch patterns
simplified display 323
solid-filled hatches 781, 788
trimming 779, 788
unbounded hatches 779
using a tool created from 84
See also hatch areas
See also hatching
hatching 770, 773–775, 781
 areas 774–775, 781
 with a solid pattern 781
dragging hatch patterns into
 drawings 770, 773
 objects 775

header rows in tables 862
height 857, 859, 862
 of rows in tables 857, 859
 of text in tables 862
helixes 448, 672
 drawing 448
 modifying 672
Help 4–5, 10–11, 20
 Autodesk website 10
 on commands 10
 for developers, starting 4
 on dialog boxes 10
 F1 key 10
 late-breaking product
 information 10, 20
 queries (natural language queries) 5
 Quick Help 11
 Readme topic 10, 20
 starting 4
 on system variables 10
 topics. *See* Help topics
 windows. *See* Help window
Help Contents. *See* Contents tab (Help window)
Help toolbar, buttons 9–10
Help topics 3–7
 displaying 3–4
 contents list 3–4
 index 3
 natural language queries vs. topic
 searches 5
 searching for words/phrases 3, 5–7
 advanced search rules 6
 basic search rules 5
 in the current topic 3
Help window 3–5, 10
 components 3
 Contents tab 3–4
 expanding (restoring) 10
 Index tab 3
 left pane 3, 10
 hiding/redisplaying 10
 Search tab 3, 5
 shrinking 10
hidden lines 989, 992
 removing 989, 992
 plotting from layout tabs 992
 plotting from the Model tab 992
hidden-line images
 See also hidden lines (in 3D objects)
Hide button (Help toolbar) 10
hiding 32, 40–41, 66, 278, 287, 779, 991
 columns, in the Layer Properties Manager 287
 command window 40–41
 control panels in dashboard 32
 dashboard 32
 hatch boundaries 779
 lines. *See* hidden lines, removing
 Model and layout tabs 66
 objects 278
 paper space objects, plotting from a layout tab 991
 tree view, in Layer Properties Manager 287
HIGHLIGHT system variable 598
highlighting 576, 578, 597
 duplicate attribute tags 576, 578
 objects 597
 visual styles and
 See also dependency highlighting
 See also selecting
highlights. *See* specular reflection
history 731–733
 composite solid history 731–732
 recording solid history 733
 removing from 3D solids 732–733
History area (QuickCalc) 404, 406–407
 changing value or expression font
 colors in 406
 clearing 406
 copying values or expressions
 from 406–407
History tab (DesignCenter) 47
Home button (DesignCenter), changing the folder 48
home folder, in DesignCenter 52
 setting to a folder with symbol library drawings 52

hook lines 817
hook lines (for dimension text) 915
horizontal 841, 909–910
dimensions
 creating 910
 extension line origins 909
 See also dimensions
 text orientation, setting 841
horizontal dimensions 876
hot grips 655
 defined 655
 multiple hot grip selection 655
hotspot angle (of spotlights) 1124
 setting 1124
HP-GL (language) 1025
HP-GL driver 1025
HP-GL format 1025
HP-GL/2 driver 1025
HP-GL/2 format 1025
HPDRAWORDER system variable 772
HPGAPTOL system variable 770, 772
HPORIGIN system variable 771
HPORIGINMODE system variable 771
HPSCALE system variable 205
hyperlink fields 830
 adding to text 830
hyphen (-) 38, 823
 dialog box suppression character 38
 no value in field display 823

|

icons 89–91, 95, 148, 1079
 Associated Standards File(s)
 icon 148
 Missing Standards File(s) icon 148
 tool palette lock icon 95
 tool palette tool icon display
 style 89–90
 tool palette tool icon updates 91
 Xref icon 1079
 exclamation point 1079
ID command 273, 392
identifying information in drawings. *See*
 drawing properties

image mapping (in rendering)
 See also material maps
image paths. *See* raster image paths
imperial units 108, 114–117, 310, 899
 converting metric units
 to/from 114–115, 117, 899
linetype definition file for 310
specifying feet and inches 116
specifying, in metric drawings 899
starting drawings based on 108
import file formats 610
 WMF files 610
importing 72, 102, 180, 214, 252, 302,
 822, 966, 969, 1037
 environment profiles 72
 layer settings 302
 layouts 214, 252
 as sheets 252
 from templates 214
 page setups 969, 1037
 for drawing sheets (in drawing
 sets) 1037
 PCP/PC2 plotter configuration
 settings 966
 into layouts 966
 RTF files 822
 text files 822
 tool palettes 102
 visual styles 180
 See also import file formats
imprinting objects on 3D solids 743–744
improving performance. *See* performance
 improvement
in-place reference editing 1096, 1098,
 1101–1103
 and attributes (of blocks) 1103
 discarding changes 1102
 inheritance of properties
 during 1101
 and nested references 1102
 and OLE objects in references 1102
 saving back changes 1101–1102
 undoing changes 1101

In-Place Text Editor 798, 800–801, 869, 872
displaying column letters/row numbers for table cell editing 869, 872
opening 800–801
transparency 798
turning off 798
inches 114–117, 404, 902–903
converting centimeters to 115, 117
converting to centimeters 114, 117
QuickCalc formats 404
specifying 116
suppressing zero values 902–903
incremental parameter values 546
incremental saves 127
indenting 811–812
paragraphs 811–812
hanging indents 812
independent base points 501–502, 509
Index tab (Help window) 3
infinite lines 450
inheritance 553, 666, 1101
layer properties, during block insertion 553
object properties 553, 666
between joined polylines 666
during block insertion 553
object properties, during in-place reference editing 1101
Input box (QuickCalc) 399, 404–407, 410, 413
appending values or expressions from the History area to 406–407
clearing 399
copying unit conversions to 410
copying values from 405
loading calculator variables into 410
pasting values to the command line 406
specifying global constants from 413
input history display 29
input properties (Property Lookup Table) 531, 533, 536
adding to lookup tables 533, 536
inscribed polygons 424–425
drawing 424–425
insert properties, of tool palette tools 91
Insert Table dialog box 857
inserting 122, 237–238, 388–391, 557, 559, 800–801, 822–826, 851, 857, 859, 864, 867–870, 872
blocks 864, 867
in table cells 864, 867
blocks. *See* inserting blocks
columns/rows in tables 859
drawing files as blocks 122, 557, 559
fields (in formulas) 872
in table cells 872
fields (in text) 823–826, 868
as placeholders 823, 826
in table cells 825, 868
formulas 868–870
in table cells 868–870
layouts 237–238
using DesignCenter 237–238
nonbreaking spaces, format code 851
points 388–391
marking equal segments on objects 388, 390–391
marking measured intervals on objects 388–389
RTF files 822–823
special characters in multiline text 800–801
tables 857
text files 822
inserting blocks 205, 388–391, 461, 465, 467, 553, 556–559, 563
at intervals on objects 388–391, 558
marking equal segments 388, 390–391
marking measured intervals 388–389
base points 461, 465
from block library drawings 558
block references and 461

with DesignCenter 558–559
drawing files as 557, 559
from drawing files 465
methods 559
object property settings 553
overview 556
system variables used for 205
with tool palettes 467, 557
with variable attributes 563
insertion point 461, 521, 523, 857
for blocks 461
for tables 857
insertion cycling 521, 523
installing 14, 56, 303, 307–308, 1058
3D DWF Publishing 1058
CAD Manager Control utility 56
color books 303, 307–308
Subscription Center 14
intensity 1123
of lights, setting 1123
interface options 65
setting 65
INTERFERE command 712
interferences 711–713
displaying 713
overlapping solids 711–712
Internet (Web) 57
retrieving content on 57
INTERSECT command 453, 708–709
intersecting 711–712
3D solids 711–712
lines. *See* intersections (of lines)
intersection edges
intersections (of lines) 409, 674
displaying the intersection of two
lines 409
multilines 674
intersections (of regions) 453–454
intervals on objects, specifying 388
inverting layer filters 295
invisible objects (in dynamic
blocks) 481, 523, 526, 529
as displayed in the Block Editor 481
making objects invisible in visibility
states 523, 526, 529
irregular 591–592
selection areas, selecting objects
in 591–592
viewports. *See* nonrectangular
viewports
irregular polygons. *See* closed polylines
ISAVEPERCENT system variable 127
islands (in hatch areas) 775, 777, 788
creating 788
removing 777
ISO (International Standards
Organization) 310, 894
dimension text placement 894
pen widths 310
ISO hatch patterns 771, 785
isolating layers 284–285
isolines
edge display
ISOLINES system variable 681
isometric 174, 443–444
circles 443–444
drawing 443–444
views 174
3D views 174
italic typeface 804
formatting multiline text in 804
See also text obliquing angle

J

jitter edge effect
jogged radius dimensions 917, 919
creating 917, 919
modifying 917
jogged segments 753, 759
section objects 753, 759
joining 196, 665–666, 668
model space viewports 196
polylines to lines, arcs, or
polylines 665–666, 668
justification 426, 802–803, 848
multiline text 802–803, 848
of multilines 426
JUSTIFYTEXT command 848

K

Kanji. *See* Big Fonts

key

 columns. *See* key values

key points (on parameters) 482, 496, 520

L

label blocks (for sheet views) 250–251, 257

 avoiding prompts when placing 251
 changing 257

landings (for dimension text) 915

landscape orientation 964, 985
 setting 964

languages 848

 checking spelling in another
 language 848

lateral tolerances 814, 903–906

 aligning vertically 904, 906
 specifying 903, 905
 stacked text in 814
 suppressing zeros 905–906

layer 0 278

layer colors 279, 286, 291, 305–306,
 581–582, 1079

 assigning 279, 286

 changing 291, 305–306

 changing color assigned to layer 581

 in xrefs 1079

 overriding 305–306, 582

Layer control (Layers toolbar) 273

layer filters 281, 294–298

 defining 294–297

 inverting 295

 nesting 298

 turning on/off 281

layer group filters 294–295, 297, 299

 defining 295, 297

 removing layers from 299

layer indexes (of xrefs) 1110–1112

 saving drawings with 1110–1112

layer linetypes 279, 291–292, 313–314,
 582–583, 1079, 1106

 assigning 279

 binding referenced linetypes 1106

 changing 291, 313–314, 582

 changing the default 292

 in xrefs 1079

 overriding 314, 583

layer lineweights 279, 291, 317, 319,
 322

 assigning 279, 317, 319

 changing 291, 322

 overriding 322

layer names 287–288, 291

 changing 291

 choosing 287

 filtering 288

layer plot styles 279, 291, 1002–1004

 assigning 279

 changing 291, 1003

 setting 1004

layer plug-in, properties checked by 143

Layer Previous, to undo layer

 changes 291

 changes that cannot be undone 291

layer properties 91, 143, 273, 279, 286–
 287, 291, 305–306, 313–314, 317,
 319, 322, 553, 582–583, 1079

 assigning 273, 279, 286, 317, 319

 colors 279, 286

 linetypes 279

 lineweights 279, 317, 319

 plot styles 279

 changing 287, 291, 305–306, 313–
 314, 322

 colors 291, 305–306

 for multiple layers 287

 linetypes 291, 313–314

 lineweights 291, 322

 plot styles 291

 colors. *See* layer colors

 copying 286

 inheritance, during block

 insertion 553

 linetypes. *See* layer linetypes

 lineweights. *See* layer lineweights

overriding 305–306, 314, 322, 582–583
colors 305–306, 582
linetypes 314, 583
lineweights 322
plot styles. *See* layer plot styles
properties checked by layer
 plug-in 143
tool palette tool overrides 91
in xrefs 1079
Layer Properties Manager 286–288, 295, 298
 assigning properties 286
 hiding a column 287
 hiding tree view 287
 selecting layers 287
 sorting layers 295, 298
layer property filters 294, 296
 defining 294, 296
layer settings 284, 290–291, 299–302
 changing 290
 editing/renameing/deleting 301
 exporting 300
 importing 302
 restoring 284, 299, 301
 saving 299–300
 undoing changes to 291
Layer State Converter (Migration Tool) 2
layer states 299
Layer Translator 153–154
layers 124, 153–156, 194, 218, 225–229, 271, 273, 277–292, 295–299, 305, 313, 317, 319, 322, 581, 593, 762, 1054, 1079, 1110
 about 277
 as object property 273
 assigning 279, 286, 290, 313, 317, 319, 322
 colors to 279, 286
 linetypes to 279
 lineweights to 279, 317, 319
 objects to other layers 290, 313, 322
 plot styles to 279
 assigning objects to other layers 305
 assigning properties to 273, 279, 286, 317, 319
 colors 279, 286
 linetypes 279
 lineweights 279, 317, 319
 plot styles 279
 changing 291, 305, 581
 for selected objects 291, 305, 581
 colors. *See* layer colors
 converting to drawing standards 153–154
 copying 286, 288
 between drawings 288
 properties 286
 copying objects to 279
 creating 287, 289
 deleting 288–289
 deleting by selecting an object 283
 displaying 155, 282
 selected layers 155, 282
 dynamically displaying 280
 filtering 280–281, 288, 296–297
 by layer property 296
 by name 296
 by selecting layers 297
 names 280–281, 288
 wildcards 280–281
 freezing 194, 225–228, 278, 283, 1110
 invisibility and 278
 in layout viewports 194, 225–228
 in paper space 228
 with selected objects 283
 xref layers 1110
frozen layers 225–226, 229, 279
 creating 226, 229
in layout viewports 227
 viewing list 227
including layer information in published DWF files 1054
indexes. *See* layer indexes
isolating 284–285
layer 0 278
 in layer property filters 295

linetypes. *See* layer linetypes
lineweights. *See* layer lineweights
live sectioning and 762
locked layer operations 593
locking/unlocking 278–279, 593
making current 279, 288
mapping to standard layers 153–154
merging 286
naming 287
nesting layer filters 298
of objects (selected) 291
 changing 291
organizing objects on 290
organizing objects with 271, 277
plot styles. *See* layer plot styles
properties. *See* layer properties
purging 155–156, 282, 290
 unreferenced layers 155–156,
 282, 290
 unused layers 290
removing from layer group
 filter 299
renaming 291–292
selecting 279, 288
settings. *See* layer settings
sorting 295, 298
standards. *See* standards (for named
 objects)
thawing 225–228
 in layout viewports 225–228
 in paper space 228
turning on/off 278, 282, 284–285
undoing changes to settings 291
unlocking 593
visibility 218, 278–279, 1079
xref-dependent layers 124
See also named objects

Layers toolbar 273, 285, 289, 291–293
 changing default linetype of a
 layer 292
 changing the layer of an object 291
 creating a new layer 289
 and the Layer control 273
 and Layer Previous 293
 removing an unused layer 289
 renaming a layer 292

turning layers on/off 285
undoing changes to layer
 settings 293
LayerWalk dialog box 280, 282
layout environment. *See* paper space
Layout Plot Area option (Page Setup dialog
 box) 954, 980
layout tabs 63, 196, 210, 212–213, 216,
 218, 243, 989, 991–992, 994
 creating new 213
 eliminating multiple tabs for sheet
 sets 243
 plotting from 989, 991–992, 994
 hiding paper space objects 991,
 994
 removing hidden lines 992
 shaded viewport plotting
 options 989, 991
 rearranging 216
 as restored from maximized
 viewports 218
 switching to the Model tab
 from 63, 196
 working on 210, 212
layout templates 236–238
 creating 237
 creating layouts with 236–237
 file name extensions 236
 saving 236, 238

layout viewports 194, 210, 217–235,
 947, 974
 accessing model space from 217
 aligning objects in different
 viewports 234
 aligning views in 233
 clipping boundaries 221, 223
 creating 219, 221, 947
 creating viewport configurations in
 layouts 221
 drawing and modifying objects
 in 217
 editing in 219
 freezing layers in 194, 225–229
 creating frozen layers 226, 229
 freezing/thawing layers
 automatically 226

locking the scale 218
maximizing 217
minimizing 217
modifying views in 218, 223
nonrectangular. *See* nonrectangular viewports
placing named viewport configurations into layouts 222
properties, modifying 222
resizing 220
rotating views in 235
scale locking 224
scaling linetypes in 231–232
scaling views in 223–224
screening (color intensity) of objects in 229–230
thawing layers in 225–228
turning on/off 230–231
view scale factor 223
viewing list of frozen layers 227
visibility of objects in 225
layouts 213–218, 221–222, 228, 236–238, 243, 252, 279, 320, 907, 946–950, 954, 956–957, 959–966, 974, 979–982, 985, 988, 990, 998–999, 1002, 1038, 1041
activating (making current) 216
activating previous (making the previous layout current) 216
applying named page setups to 947–948
assigning plot style tables to 998, 1002
copying 215
creating 213–215, 236–237, 965
with the Layout wizard 213, 215, 965
with templates 236–237
creating viewport configurations in 221
deleting 215
dimensioning in 907
drawing orientation 964, 985
duplicating 215
freezing layers in paper space 228
importing 214, 252
as sheets 252
from templates 214
importing PCP/PC2 settings into 966
initial display 946
initialized layouts 947
inserting 237–238
using DesignCenter 237–238
lineweight 320
display in 320
model layouts 1038, 1041
including in drawing sets 1038, 1041
moving objects between model space and paper space 216
multiple 243
as sheets from single drawing files 243
paper size 950, 981–982
placing viewport configurations into 222
plot area 954, 980–981
plot offset 956
plot options 962–963
plot scales 957, 985
plot style tables 959–960, 999
creating 960
editing 960
previewing the effects of 999
selecting 959
plot styles, displaying 961
plotting 217
renaming 215
reusing 236
saving page setup plot settings to 979
selecting all 217
selecting printers/plotters for 948–949
settings 213
shaded viewport plotting options 962–963, 988, 990
thawing layers in paper space 228
viewing objects in 974

visibility of layers in 218, 279
LAYUNISO command
LEADER command 843
leader lines. *See* leaders (leader lines)
leader objects. *See* leaders (leader lines)
leader text 795, 818, 820–821, 847
 aligning 821
 associativity with leaders 795, 818
 creating 820
 editing 847
 offsetting 820
 text boxes for 821
leaders (leader lines) 795, 817–820, 846, 890, 937
 annotations. *See* leader text
 associativity with leader text (mtext objects) 795, 818
 associativity with objects arrowheads are attached to 795
 automatic 795, 890
 creating 818–820
 creating geometric tolerances with 937
 dimension line\=associated 795, 817
 vs. leader lines (automatic) 795
multiple leaders 820
resizing 846
scaling 846
spline leaders 819
stretching 846
least material condition value (L/LMC) 938
left mouse button. *See* pick button (mouse)
LENGTHEN command 638
lengthening objects 638, 641
lens length of cameras 187
 defined
 specifying 187
lettered lists 805–808, 810
 creating as you type 805, 808
 formatting multiline text as 806
 moving items down a level 810
 removing list formatting from 807
 separating items from 810
letters 799, 804
 selecting 799, 804
light glyphs 1118, 1121–1122
lighting 179, 1117–1124, 1127–1129
 about 1117–1118
 adding lights 1118
 color 1123, 1127, 1129
 converting older versions of drawings
 deleting lights
 distant. *See also* distant lights
 distant 1118, 1120–1121
 face styles and
 geographic locations and 1127–1128
 hotspots and falloff 1124
 intensity 1123
 light glyphs 1118, 1121–1122
 light tools 1121
 listing 1122
 locations 1122–1123
 point lights 1118–1120, 1127
 positioning 1127
 properties 1122
 See also attenuation of light
 See also reflection (of light)
 selecting
 shadows 1123
 simulating sunlight 1118, 1127–1129
 about 1118, 1127
 about lighting 1118
 brightness 1128
 sun angle 1129
 sun color 1129
 turning sun on/off 1127
spotlights 1118–1120, 1127
types of 1118
visual styles and 179
ambient. *See also* ambient light
point. *See also* point lights
spotlights. *See also* spotlights
lightness
 See also brightness
limits 904–905
tolerances 904–905

See also drawing limits
LIN (linetype library) files 310
line breaks 865
 in table cells 865
LINE command 359, 418
 dimensional input 359
line end styles 1015–1016
 assigning 1015–1016
line join styles 1016
 assigning 1016
line segments 316, 646, 650
 chamfering 650
 filleting 646
linetypes on short segments 316
 See also lines
line spacing, multiline text 813
line text. *See* single-line text
line widths. *See* lineweights
Linear Array parameter set 518
linear dimensions 876, 909, 932
 creating 909
 definition points 932
 rotated. *See* rotated dimensions
 types 876
linear grips 520, 561
Linear Move Pair parameter set 518
Linear Move parameter set 518
linear objects. *See* lines
linear parameters 482, 484, 488, 518,
 520, 539, 543, 546–548
 actions supported by 482, 484
 adding to dynamic block
 definitions 488
 Chain Actions property 484, 539,
 547–548
grip numbers 520
grip type 482
incremental values 546
minimum and maximum
 values 546
sets 518
value sets 543, 546
Linear Stretch Pair parameter set 518
Linear Stretch parameter set 518
linear unit conventions 115
 setting 115
lined grid 375
lines 189, 196, 318, 376, 387, 408–409,
 418–419, 423, 434–435, 644, 665–
 666, 863
centerlines. *See* centerlines
construction. *See* construction lines
converting polylines into 423
creating solids from
dimension. *See* dimension lines
dimensioning. *See* linear dimensions
displaying the angle of a line 408
displaying the intersection of two
 lines 409
drawing 318, 376, 387, 418–419,
 434–435
 in model space 318
 tangent to arcs 434–435
 using direct distance entry 387
 using orthogonal locking 376
drawing, in viewports 196
extension. *See* extension lines
filleting lines with polylines 644
freehand. *See* freehand lines
hidden. *See* hidden lines
intersecting. *See* intersections (of lines)
joining to polylines 665–666
motion path animations and 189
multilines. *See* multilines
of text. *See also* single-line text
perpendicular lines 376
polylines. *See* polylines
segments. *See* line segments
styles. *See* linetypes
table gridlines 863
weights. *See* lineweights
Linetype control (Properties
 toolbar) 273, 313
linetype definition files 311
 listing linetypes in 311
linetype scale 315–316, 426
 changing 315–316
 globally 316
 for selected objects 315
Current Object Scale 315
Global Scale Factor 315
multiline scale and 426

setting (making current) 316
linetypes 205, 231–232, 271, 273, 278–279, 291–292, 309–317, 430, 553–554, 581–583, 594, 609, 1014–1015, 1106
assigning 279, 312, 554
 to layers 279
 to objects 312, 554
assigning, to objects 273
binding referenced linetypes 1106
changing 291, 313–314, 582–583
 layer linetypes 291, 313–314,
 582
 object linetypes 313–314, 583
CONTINUOUS linetype 278
current linetype 312
definition files 311
 listing linetypes in 311
deleting 310
displaying 316–317
 on polylines 316–317
 on short segments 316
filtering selection sets based on 594
for freehand lines 430
hardware linetype use 309
in blocks 553, 581
for layers. *See* layer linetypes
listing 310–311
 definition file linetypes 311
 loaded linetypes 310
loading 292, 310
for objects. *See* object lineweights
plot style linetypes 1014–1015
purging 311, 609
scale. *See* linetype scale
scaling 205, 231–232, 315, 1014–1015
 in layout viewports (paper
 space) 231–232
 non-ISO linetypes 1015
setting (for new objects) 273
setting (making current) 312, 554
standards. *See* standards (for named
 objects)
unloading 311
See also named objects

Lineweight control (Properties
 toolbar) 273
lineweight scale 319, 958
 displaying 319
 setting 958
 for layouts 958
Lineweight Settings dialog box 318
 accessing 318
lineweights 271, 273, 279, 291, 317–
 322, 324–325, 553, 594, 863,
 866, 958, 990, 993, 1014
 assigning 279, 317–319, 321
 to layers 279, 317, 319
 to objects 318, 321
 to polylines 318
 assigning, to objects 273
 changing 291, 322
 layer lineweights 291, 322
 object lineweights 322
current linewidth 321
default value 318
displaying 318–320
 in model space 318–319
 in paper space layouts 320
 scale 319
 turning off 320
 displaying at current setting 321
filtering selection sets based on 594
in blocks 553
for layers. *See* layer lineweights
for objects. *See* object lineweights
objects not displaying 318
plot style lineweights 1014
plotting 318, 990, 993
 with custom values 318
in previous release drawing
 objects 321
and regenerating drawings 320
scaling 958
setting (for new objects) 273
setting (making current) 318, 321
for table cell borders 866
table gridlines 863
turning on/off 324–325
values 318
See also named objects

link path names. *See* link templates
linking 189
 cameras to paths 189
 See also attaching
links (in databases)
 See also database links
LIST command 273
listing 36, 275, 310–311, 881, 1107
 database information 275
 dependent named objects (in
 xrefs) 1107
dimension style overrides 881
dimensioning system variables 881
linetypes 310–311
 in definition files 311
 loaded linetypes 310
system variables 36
xrefs 1107
lists in multiline text 805–810
 creating as you type 805, 808–809
 formatting multiline text as 806
 moving items down a level 810
 removing list formatting from 807
 separating items from 810
live sectioning 750–751, 758, 761–762
 about 751, 761
 cameras and 758
 cutaway geometry 762
 defined 750
 features 762
 turning on/off 762
LMC least material condition
 symbol 938
loading 51, 125, 292, 310
 DesignCenter content area 51
 geometry into partially open
 drawings 125
 linetypes 292, 310
 xrefs
 on demand. *See* demand loading
locating 357
 points 357
location
 cameras
lock icon 95
 in tool palettes 95
lock icons 266–267, 359
 in dimensional input 359
 for sheet sets in Sheet Set
 Manager 266–267
 false 267
locked layers 279, 593
 defined 279
 operations possible on 593
locking 11, 65, 67, 218, 278–279, 380,
 476, 593
 angles, specifying points by 380
 Block Editor 476
 layers 278–279, 593
 the layout viewport scale 218
 Quick Help (Info palette) 11
 toolbars 65, 67
 windows 65, 67
 See also orthogonal locking
 See also scale locking
LOFT command 699, 701
lofting objects 729–730
 3D objects 729–730
 surfaces 729–730
log files 1109
 for xrefs 1109
 turning on/off 1109
logo screen, not displaying on startup 75
lookup actions 515–516, 518, 531
 adding to dynamic block
 definitions 515–516, 518,
 531
 parameters supported by 516
lookup grips 520, 523, 531, 534, 561
lookup parameters 482, 484, 495, 518
 actions supported by 482, 484
 adding to dynamic block
 definitions 495
 grip type 482
 sets 518
lookup properties (Property Lookup
 Table) 531, 533–534, 536–537
 adding to lookup tables 533, 536
 allowing for reverse lookup 534
 making read only 537
 overview 531
Lookup Set parameter set 518

lookup tables (for dynamic blocks) 515, 531–533, 535–538
adding input properties and lookup properties 533
adding input properties to 536
adding lookup properties to 536
adding lookup values to 537
assigning data to dynamic blocks 531
checking for errors in 531, 538
creating 515, 532, 535
opening 538
value specification rules 538
lookup values (Property Lookup Table) 537–538
adding to lookup tables 537–538
LTSCALE system variable 315
LWDEFAULT system variable 318

M

M maximum material condition symbol 938
magnet (AutoSnap lock) 368
magnifying views. *See* zooming
main window. *See* application window
major grid lines 372, 375
manufacturer content 54–56, 58
accessing 54
downloading 55, 58
mapping 153–154, 430
2D images onto 3D object surfaces.
See image mapping
coordinate systems on paper to the WCS 430
layers to standard layers 153–154
margins in table cells 862
markers 368, 383–384, 389–390
for object snaps 368
for point objects 389–390
for tracking points 383–384
marking 388–391
equal segments on objects 388, 390–391
measured intervals on objects 388–389

markups (markup objects) 455
highlighting with revision clouds 455
masking objects 790
master drawings, creating 1078
material condition symbol 936, 938
material conditions (of features) 938
materials
texture. *See* textures
math expressions (in QuickCalc). *See* expressions (in QuickCalc)
math functions (in QuickCalc) 400
using 400
mathematical expressions 403, 414
evaluating in dialog boxes 403, 414
MaxArray system registry variable,
setting 623
MaxHatch system registry variable 772, 787, 789
setting 772, 787, 789
maximum material condition value (M/MMC) 938
measurement units. *See* units of measurement
measuring intervals on objects 388–389
mechanical blocks, accessing collections of 58
memory
memory allocation
memory functions (in QuickCalc). *See* QuickCalc memory
menu bar 25
menu grips 755
menus 25–26, 69, 131
choosing options 25–26
custom menus 131
in workspaces 69
for workspaces
merging 286, 858–859
cells in tables 858–859
layers 286
meshes (faceted surfaces) 713–717, 719–721
overview 713
polyface (polygon). *See* polyface meshes

predefined meshes 717
specifying 714
types 713–717, 719–721
metadata options for published DWF files 1050
metafiles. *See* WMF files
metric units 108, 114–115, 117, 310, 899
converting imperial units to/from 114–115, 117, 899
linetype definition file for 310
specifying, in English unit drawings 899
starting drawings based on 108
Microsoft Office 16
 Vault Office Add-in 16
Migration Tools 2
minor grid lines 372
minus sign (-), tree view level display button 47
mirroring 629–631, 656, 659, 663, 842
 objects 629–631
 in 3D 630–631
 objects, with grips 656, 659, 663
 text 629, 842
 See also flipping
MIRRTEXT system variable 629, 842
Missing Standards File(s) icon 148
mistakes, correcting. *See* undoing actions
MMC maximum material condition symbol 938
model layouts 1031, 1038, 1041
 including with drawing sheets 1031, 1038, 1041
model space 63, 184, 204, 210–212, 216–218, 318–319, 762, 847, 906–907
 accessing from layout viewports 217
 cameras in 184
 converting text height between paper space and 847
 dimensioning in 906–907
 for plotting in 906
 for plotting in paper space 907
 lineweight display in 318–319
 live sectioning and 762
moving objects between model and paper space 212, 216
switching between paper space and 218
switching from a layout tab to 63
uses 211
working in (on the Model tab) 204, 210–211, 217
See also Model tab
model space viewports 194–197
 coordinate systems in 195
 cycling through 196–197
 drawing lines in 196
 joining 196
 making current 196–197
 restoring 196
 subdividing 195
 uses 194
Model tab 63, 196, 205, 210–212, 217–218, 981, 985, 989, 991–992
 activating 212
 layer visibility settings 218
 plot scale 985
 plotting from 205, 981, 989, 991–992
 removing hidden lines 992
 shaded viewport plotting options 989, 991
 switching from a layout tab to 63, 196
 working on 210–211, 217
model viewports 166, 346, 354
arrangements. *See* viewport arrangements
multiple 166, 346, 354
 in 3D views 346, 354
UCSs in 346
using the Aerial View window with 166
modifying 95, 218–219, 223, 249, 264, 549–550, 573–577, 580, 583, 611, 665, 667, 672, 674, 785, 787–789, 799, 832, 841–844, 857, 884, 886, 917, 925, 928, 931–933, 947, 953, 968, 984, 1032,

1036–1037, 1039, 1042, 1086,
1095–1096
2D solids 788
action selection sets 550
associative dimensions 931
block attributes 576–577, 580
 applying changes to block
 references 577
 in block definitions 576–577
 in block references 577, 580
block definitions 573–574, 583,
 1095–1096
block descriptions 574–575
blocks
 See also in-place reference editing
dimension 928
 text 928
dimension lines 884
dimensions 925
donuts 788
double-click editing 611
dynamic block definitions 549
exploded dimensions 933
extension lines 886
gradient fills 785, 789
hatch boundaries 788
hatch patterns 788
hatches 787
helixes and spirals 672
jogged radius dimensions 917
layout viewports 219
multiline text 799, 843–844
multilines 674
named page setups 968
nonassociative dimensions 932
objects 611
 methods 611
page setups 947, 1032, 1036–1037,
 1039, 1042
 for drawing sheets in drawing
 sets 1032, 1036–1037,
 1039, 1042
paper size 953, 984
polylines 665, 667
printable area of a drawing 984
project names 1086
search paths 1086
sheet selections 264
sheets 249
solid-filled areas 788
tables 857
text objects 841–843
 single-line text objects 842–843
text styles 832
 and multiline text character
 formatting 832
tool palette tools 95
views 218, 223
 in layout viewports 218, 223
wide polylines 788
See also editing
monochrome faces
motion path animation 189–191
 about 189
 linking cameras to paths 189
 previewing 191
 recording 191
 saving 191
 settings 190
Motion Path Animation dialog box 190
mouse buttons
 See also pick button (mouse)
move actions 498–500
 adding to dynamic block
 definitions 498–499
 Distance Type property 499–500
 parameters supported by 498
move grip tool 735, 738
moving 95, 188, 212, 216, 283, 339,
 503, 612–615, 656, 658, 730, 736–
 738, 740–741, 843–844, 857, 928,
 1127–1128
3D solid subobjects
axis of revolution/rotation 730
cameras 188
dimension text 928
geographical location of
 models 1128
grip tools 737
lights 1127
multiline text 843–844

objects 212, 216, 283, 503, 612–615, 736, 738, 740–741
between model and paper
 space 212, 216
by stretching 503, 613, 615
constraining movement 738, 740–741
from paper space to model
 space 614
grip tools and 736, 738
point method 612–613
by relative displacement 612, 614
 to other layers 283
objects, with grips 656, 658
tables 857
tool palette tabs 95
UCSs 339
 See also converting
mtext. *See* multiline text
MTEXTED system variable 850
multi-sheet DWF files 1051–1052
 default names 1052
 specifying 1052
name prompt 1052
 specifying 1052
specifying 1051
multi-sheet PDF files 1052
 naming 1052
multiline intersections 674, 676
 modifying 674, 676
multiline objects. *See* multilines
multiline scale 426
 and linetype scale 426
multiline styles 426–427, 674, 676
 creating 427
 editing 674, 676
multiline text 794, 798–804, 806, 811–814, 832, 838–841, 843–845, 848, 851–852
 aligning 802–803, 852
 vertical alignment format
 codes 852
boldfacing 804

character formatting 832
 as affected by modifying text
 styles 832
character spacing, format code 852
coloring 799, 804, 851
 format code 851
creating 798–799, 843, 851
 in alternate text editors 851
 without a bounding box 843
editing 794, 843–844, 851
 in alternate text editors 851
finding 844–845
fonts, changing 804
format codes 852
formatting
 in alternate text editors 851
 as lists 806
 See also text styles
height 804, 838–839, 852
 changing 804
 format codes 852
 setting 838–839
indenting 811–812
 hanging indents 812
inserting special characters in 800–801
italicizing 804
justification 802–803, 848
line spacing 813
line width 798–799
 defining 798–799
lists 806
 delimiter characters 806
 prohibited characters for
 bullets 806
modifying (changing) 799, 843–844
moving 843–844
as objects. *See* multiline text objects
obliquing angle 840, 852
 format code 852
orientation 841
overlining, format codes 851
paragraph end format code 852
properties 799
replacing 844–845
selecting 799, 804

smoothing fonts 802
stacked 814, 852
 format codes 852
styles
 not affecting 832
 overriding 803
 See also text styles
tabs, setting 811
underlining 804, 851
 format codes 851
wide text, format code 852

Multiline Text Editor 811–812, 833, 850
 fonts displayed 833
 ruler 811–812
multiline text objects 794–795, 798–799, 818
 background options 798
 defining 798–799
 leader associativity with 795, 818
 See also multiline text

multilines 426–427, 674–676, 879
 adding vertices 674
 and associative dimensioning 879
 deleting vertices 674
 double lines. *See* double lines
 drawing 427
 editing commands not for 675
 end points 675
 intersections 674, 676
 justification of 426
 modifying 674
 properties 426
 scale 426
 vertices 674–675
 See also multiline styles

multiple copies (of objects) 660–661
 making, with grips 660–661

multiple drawings 126, 147, 152
 checking for standards
 violations 147, 152
 copying between 126
 displaying 126
 switching between 126

multiple hot grip selection 655

multiple layers 287
 changing properties 287
 selecting 287

multiple layout viewports, creating 220

multiple layouts 243
 as sheets from single drawing files 243

multiple object snaps. *See* running object snaps

multiple objects 590
 selecting 590

multiple sets of objects 642, 647, 652
 chamfering 647, 652
 filleting 642, 647

multiple text objects, scaling 847

multiple UCSs 346

multiple viewports (model viewports) 166, 346, 354
 in 3D views 346, 354
 UCSs in 346
 using the Aerial View window with 166

multiple-line objects. *See* multilines

MVIEW command, creating
 nonrectangular layout
 viewports 220

MVSETUP command 234, 236
 aligning objects in different layout viewports 234
 rotating views 236

My Documents folder 122, 131–132
 specifying the search path 131–132

MYDOCUMENTSPREFIX system variable 131

N

name conflicts in xrefs 1105
named
 objects
 dependent. *See* dependent named objects (in xrefs)
 standards. *See* standards, for named objects

page setups
 See also page setups

queries. *See* stored queries

viewport arrangements. *See* viewport arrangements
named groups 602, 604
 deleting 604
named layer states 299
 restoring 299
 saving 299
named object selection filters 594, 596
named objects 123, 142–144, 147–149,
 607, 609, 832, 1107
 availability in partially open
 drawings 123
 defining properties for 144
 fixing objects with standards
 violations 143, 147–149
names, special characters for 832
nonstandard 142–143, 148
 fixing 143
purging 607, 609
 linetypes 609
renaming 1107
types for which standards can be
 created 142
named page setups 946–948, 966–971,
 979
 applying 947–948, 966–967
 to other layouts 947–948
 applying as override when publishing
 sheets 971
 applying to multiple layouts 946
 applying to sheet created from
 scratch 971
creating 947, 967, 979
deleting 969
importing 969
importing from a drawing 946
modifying 968
renaming 970
using with sheet sets 971
named plot style tables 975, 996–997,
 1002, 1004, 1007
and color mapping tables 1007
converting color-dependent plot style
 tables to 997
converting drawings from
 color-dependent tables
 to 996–997
converting drawings to
 color-dependent tables
 from 997
file name extension 975
managing plot styles in 1004
plot styles in 1002
named plot styles 975, 996–997, 1004–
 1007
converting drawings from
 color-dependent styles
 to 996–997
converting drawings to
 color-dependent styles
 from 997
copying 1005
creating (adding) 1005, 1007
deleting 1006–1007
description, changing 1005
managing 1004
predefined styles 1007
renaming 1006–1007
named UCSSs 342, 350
 restoring 342, 350
named views 167–170, 758
 deleting 170
 renaming 169
 restoring 168–169
 saving 167–168
 settings saved 167
 section objects and 758
naming 128, 168–169, 187, 198, 287,
 296, 757, 832
cameras 187
drawings 128
 for international work 128
layers 287
objects, using wild-card
 characters 296
section objects 757
text styles 832
viewport arrangements 198
views 168–169
See also renaming

- natural language queries (in Help) 5–7
 tips for good results 6
 Web searches 7
- navigating 11, 36
 in the command window 36
 Quick Help (Info palette) 11
- NEAR operator, Help topic searches 6
- nested blocks 555, 572, 1093, 1102
 clipped 1093
 defined 555
 in attribute extraction files 572
 in-place reference editing and 1102
- nested layer filters 298
- nested xref paths 1089
- nested xref paths, changing 1104
- nested xrefs 1088–1089, 1093, 1102,
 1104
 clipped 1093
 in-place reference editing and 1102
 paths 1089
 paths, changing 1104
- Network Administrators Guide*, accessing 2
- New Table Style dialog box 861
- no path option, for xrefs 1083
- nonassociative dimensions 878, 932,
 935
 converting associative dimensions
 into 935
 converting into associative
 dimensions 935
 filtering selections 878
 modifying 932
- nonassociative hatches 772, 788
- nonbreaking spaces, inserting, format
 code 851
- nonnumeric characters, stacking 815
- nonrectangular layout viewports 220
 creating 220
 suppressing boundary display 220
- nonstandard named objects 142–143,
 148
 fixing 143
- nonsystem plotter/printer drivers 983,
 1019–1022, 1024–1025
 custom paper sizes 983
 DXB driver 1020
- HP-GL driver 1025
HP-GL/2 driver 1025
PDF output 1022
PostScript driver 1024
raster driver 1021
- NORMAL plot style 1004, 1008
- normals (of surfaces) 730
- NOT operator, Help topic searches 6
- notification 28, 146, 148, 150, 1079,
 1083, 1090
 of attached xrefs 1079, 1083
 relocated xrefs 1083
 of changed xrefs 1090
 from services 28
 of standards violations 146, 148,
 150
 turning on and off 148, 150
- noun/verb selection of objects 589, 597–
 598
- Number of Grips property 520
- numbered lists 805–808, 810
 and creating formatted lists 805
 creating as you type 805, 808
 formatting multiline text as 806
 moving items down a level 810
 removing list formatting from 807
 separating items from 810
- numeric characters, stacking 815
-
- object classifications 594
- object colors 273, 279, 286, 302, 304–
 306, 553–554, 582, 1010–1011
 assigning to layers 279, 286
 block insert settings 553
 changing 305–306, 582
 overriding layer colors 305–
 306, 582
 converting to grayscale 1010–1011
 setting (for new objects) 273, 302,
 304
 setting (making current) 554
- See also* colors

object linetypes 273, 279, 309, 312–314, 553–554, 583
assigning to layers 279
block insert settings 553
changing 313–314, 583
 overriding layer linetypes 314, 583
setting (for new objects) 273
 setting (making current) 312, 554
object linewidths 273, 279, 317–319, 321–322, 553, 990, 993
assigning to layers 279, 317, 319
block insert settings 553
changing 322
 overriding layer
 linewidths 322
plotting 990, 993
setting (for new objects) 273
 setting (making current) 318, 321
object linking and embedding. *See* OLE
object plot styles 279, 1002, 1004
 assigning to layers 279
 changing 1002
 setting 1004
 See also plot styles
object properties 271, 273–276, 279, 286, 302, 304–305, 312–314, 317–319, 321–322, 475, 553–554, 582–583, 594, 825, 1028, 1101
 assigning to layers 273, 279, 286, 317, 319
 colors 279, 286
 linetypes 279
 linewidths 279, 317, 319
 plot styles 279
 assigning to objects 273, 302, 304
 colors 302, 304
 changing 273, 275, 305, 313–314, 322, 582–583
 colors 305, 582
 linetypes 313–314, 583
 linewidths 322
 colors. *See* object colors
copying 276
 suppressing specific
 properties 276
display properties 322
displaying 273–274, 475
 in the Block Editor 475
methods 273
with the Properties palette 274
displaying in fields 825
filtering selection sets based on 594
floating properties 553
in blocks 553
 insert settings 553
inheritance 553
 during block insertion 553
inheritance, during in-place reference
 editing 1101
listing database information 275
setting (for new objects) 273, 302, 304
 colors 302, 304
setting (making current) 312, 318, 321, 554
 colors 554
 linetypes 312, 554
 linewidths 318, 321
specifying block-related properties for
 publishing in DWF
 files 1028
object selection 589, 593–594, 596–599, 602–603
 area transparency 597
 customizing 596
 cycling 589, 602
 in groups 602
 filtering 597
 filters 594, 596
 group selection, toggling 602–603
 highlighting
 previewing effects. *See* selection
 previewing
 turning off 598
 methods (default options) 598–599
 changing 599
 preventing 593
 previewing effects 597–599
 turning on/off 599
Object Selection Filters dialog box 594

object snap 365–369
 aperture box 368
 magnet 368
 marker 368
 menu 367
 customizing 367
 displaying 367
 overriding 369
 setting to ignore hatch objects 367
 specifying 365
 tooltips 368
 using in 3D 366
Object Snap shortcut menu 365
 displaying 365
object snap tracking 383–385
 changing AutoTrack settings 385
 settings 384
 tips 384
 tracking points 383–384
 turning on/off 385
object snaps
 running. *See* running object snaps
 visual aids. *See* AutoSnap
 See also running object snaps
objects 80–81, 94, 127, 129, 212, 216–217, 225, 229–230, 234, 273, 275–276, 278–279, 283, 286, 290–291, 302, 304–305, 313, 322, 325, 366, 379, 388–391, 393, 417, 455, 469, 478, 558, 581, 588–589, 591–592, 595, 603–604, 607–622, 627–632, 634–642, 645–652, 654–663, 696–699, 701, 704–706, 728–730, 775, 790, 974, 1063, 1099–1100
3D objects. *See* 3D objects
adding to groups 603
adding, to selection sets 595
aligning 234, 618–619
 in different layout
 viewports 234
assigning properties to 273, 302, 304
 colors 302, 304
assigning to other layers 290, 305, 313, 322
breaking 652, 654
calculating 393
 area/perimeter/circumference 393
chamfering 647–651
 by setting the distances 648, 650
 by specifying length and angle 649, 651
 without trimming 651
changing layers of 581
colors. *See* object colors
converting to revision clouds 455
copying 276, 279, 609–610, 620–622, 660–662
 in arrays 622
 to the Clipboard 609–610
 with grips 660–661
 multiple copies 621, 660–661
 with offset snaps 660, 662
 point method 620–621
 properties between 276
 properties between, suppressing specific properties 276
 by relative displacement 621
 with rotation snaps 660, 662
 to other layers 279
creating 729–730
 by lofting 729–730
creating by lofting 701
creating solids or surfaces from 704–706
creating tools from 80–81, 94
creating with tool palette tools 80
cutting to the Clipboard 608–610
cycling through 589
database information, listing 275
displaying in Block Editor 478
draw order 325
 changing 325
drawing 80, 217, 379, 417, 469, 697–698, 704
 by revolving 704
 by sweeping 697–698
 creating tools for 80
 for dynamic blocks 469
 in layout viewports 217

using polar distances 379
using polar tracking 379
editing 611, 655
 methods 611
 with grips 655
erasing 607–608
exploding. *See* exploding compound objects
exporting 1063
 to 3D DWF files 1063
extending 632, 634–636
 in 3D 635–636
 while trimming 632
extruding into solids or surfaces 696
filleting 642, 645–646
 in 3D 645
 without trimming 646
hatching 775
hiding 278
inserting blocks at intervals on 388–391, 558
 marking equal segments 388, 390–391
 marking measured intervals 388–389
intersecting. *See* intersecting objects
invisible. *See* invisible objects (in dynamic blocks)
layers of selected objects 291, 305
 changing 291, 305
lengthening 638, 641
linetypes. *See* object linetypes
lineweights. *See* object lineweights
lofting to create solids 699, 729–730
markups. *See* markups (markup objects)
masking (covering) 790
measuring intervals on 388–389
merging to another layer and deleting the original layer 286
mirroring 629–631, 656, 659, 663
 in 3D 630–631
 with grips 656, 659, 663
in model space. *See* model space objects
modifying 611
 methods 611
moving 212, 216, 283, 612–615, 656, 658
 between model and paper space 212, 216
 from model space to paper space 614
point method 612–613
by relative displacement 612, 614
by stretching 613, 615
to current layer 283
 with grips 656, 658
named. *See* named objects
noun-verb selection 589
offsetting 627–629, 660, 662
 in copying with grips 660, 662
distance method 628
point method 629
polylines or splines 628
OLE. *See* OLE objects
in paper space. *See* paper space objects
pasting 610–611
 into drawings 610–611
points. *See* point objects (points)
properties. *See* object properties
reassigning to different layers 290
removing 588–589, 591–592, 603, 1099–1100
 from working sets 1099–1100
 from groups 603
 from selection sets 588–589, 591–592
reordering in groups 603–604
reshaping 638
resizing 638
revolving 729–730
rotating 615–618
 in 3D 616
 to an absolute angle 616–617
 around an axis 618
 by dragging 615
 by a specified angle 615
rotating, with grips 656, 658
saving selected objects 127, 129

scaling 639–641, 656, 659
 by reference 640–641
 by scale factor 639–640
 with grips 656, 659
screening (color intensity) 229–230
 in layout viewports 229–230
selecting. *See* selecting objects
snapping to points on 366
stretching 639–640, 656–658
sweeping to create solids 698, 728
text. *See* text objects
trimming 632, 634–637, 646, 651
 in 3D 635, 637
 chamfering without 651
 while extending 634
 filleting without 646
viewing in layouts 974
visibility 225, 278
 layer control 278
 in layout viewports 225
See also geometry
oblique 915
 extension lines 915
obliquing angle (text) 840, 852
 setting 840, 852
 format code (multiline
 text) 852
obscured edges
offset snaps, copying objects with 660,
 662
offsetting 388, 450, 627–629, 660, 662,
 820, 985
 construction lines 450
 drawings (on paper) 985
 leader text 820
 objects 627–629, 660, 662
 in copying with grips 660, 662
 distance method 628
 point method 629
 points, from temporary reference
 points 388
polylines 628
splines 628
OLE objects 822, 1102
 in blocks 1102
 pasted text as 822
 in xrefs 1102
one-color gradient fills 784
 creating 784
online content (for drawings)
 (DesignCenter Online) 54–58
 accessing 54, 56–57
 browsing 57
 downloading 55, 58
 previewing 55, 58
 searching for 54, 57–58
 types 56
OOPS command 605
OPEN command 123
 Partial Open option 123
Open Drawings tab (DesignCenter) 47
opening 17, 50, 53, 122–124, 151, 244–
 245, 249, 253–254, 274, 472–474,
 538, 549
block definitions 472, 549
blocks 473–474
Communication Center 17
drawing files as blocks 474
drawings
 authenticity check 122
 with DesignCenter 50, 53
 partially 123–124
 See also starting drawings
 within sheet sets 249, 253–254
 in Windows Explorer 122
dynamic block definitions 549
dynamic blocks 474
lookup tables (for dynamic
 blocks) 538
Properties palette 274
Sheet Set Manager 244
sheet sets 245
 standards check files 151
operators. *See* query operators
optimizing program performance. *See*
 performance improvement
Options dialog box 65, 76, 131
 environment settings,
 precedence 76
Files tab 131
 setting interface options 65
OR operator, Help topic searches 6

ordinate dimensions 922–923, 932
automatic 922
creating 923
definition points 932
See also dimensions
organizing 98, 119, 246, 277, 290, 467
blocks 467
objects on layers 277, 290
sheet sets 119, 246
tool palettes 98
orientation 841, 964, 985
of drawings 964, 985
text 841
origin 328, 339
coordinate systems
See also UCS origin
Ortho mode 376–377
locking (Ortho mode) 376
and polar tracking 376
drawing lines 376
overrides 376
turning on/off 376
and polar tracking 377
orthogonal extension lines 923
orthographic projections. *See* parallel projections (of 3D views)
orthographic sectioning 751, 753
OSNAP button (status bar) 365
turning running object snaps on/off 365
OSNAPZ system variable 366
Otrack button (object snap tracking) 383
outlines. *See* lists in multiline text
output directory for published DWF files 1051
changing the default 1051
overall dimension scale, setting 907
overhang edge effect
overlapping objects 325, 711–712
3D solids 711–712
draw order 325
changing 325
overlaying 1089
xrefs 1089
overlaying xrefs 1081, 1088
vs. attaching xrefs 1088
with DesignCenter 1081
overlays. *See* layers
overlining multiline text, format codes 851
override keys 369, 378
overrides. *See* dimension style overrides
overriding 243, 262, 369–370, 373, 375, 645, 648, 964
chamfer distances 648
the fillet radius 645
object snaps 369–370
running object snaps 369–370
page setups 243, 262, 964
for sheet sets 243, 262
the snap (grid snap) 373, 375
overriding page setups 975

P

page settings. *See* page setups
Page Setup dialog box 954
plot area options 954
Page Setup Manager 947
displaying with new layouts 947
page setup override templates 262
page setup overrides 1046
publishing sheets (in sheet sets)
using 1046
page setup overrides file 243, 1043, 1046
page setups 214, 243, 262, 947–948, 964, 969, 971, 975, 978–979, 1032, 1036–1037, 1039, 1042, 1046
for the current layout 214
for sheet sets 243, 262
overriding 243, 262
importing 969, 1037
for drawing sheets (in drawing sets) 1037
modifying 947, 1032, 1036–1037, 1039, 1042
for drawing sheets in drawing sets 1032, 1036–1037, 1039, 1042

named 947–948, 969, 971, 979
applying as override when
publishing sheets 971
applying to other layouts 947–
948
applying to sheet created from
scratch 971
creating 947, 979
importing 969
overriding 243, 262, 964, 975
for sheet sets 243, 262
plot settings 964, 978–979
plotting with 978
saving 979
specifying 978
publishing sheets (in sheet sets) using
page setup overrides 1046

page size. *See* paper size

palettes 68

- Auto-hide, turning on/off 68
- See also* color palettes
- See also* Properties palette
- See also* tool palettes

pan box. *See* view box (Aerial View window)

PAN command 160

panning 69, 160, 162, 164–166

- 3D view
- with the Aerial View window 164–
166
- by dragging 162
- dynamic (Realtime option) 160
- by specifying points 162
- turning on animation for 69

paper 957, 985–987

- offsetting drawings on 985
- scaling drawings to fit 957, 986–987

paper drawing sets 1037–1038

- creating and publishing 1037–1038

paper image

- See also* printable area

paper sheet sets 1043

- publishing 1043

paper size 950–953, 981–986

- custom sizes 951–953, 981, 983
- adding 951–952, 981, 983

deleting 953

editing 952, 983

modifying 953, 984

and plot scale 985–986

setting 950, 981–983

- for layouts 950, 981–982
- for plotters 981, 983

paper space 173, 210–212, 216, 218,
228, 232, 320, 350, 465, 847, 907

- converting text height between model
space and 847
- dimensioning in model space for
plotting in 907
- freezing layers in 228
- lineweight display in 320
- moving objects between model and
paper space 212, 216
- objects 465
- copying between drawings 465

objects in. *See* paper space objects

scaling linetypes in 232

- switching between model space
and 218

thawing layers in 228

UCS limitations 350

uses 211

viewing commands not available
in 173, 350

views 173

working in (on a layout tab) 210,
212

paper space objects 212, 216, 991, 994

- hiding, plotting from a layout
tab 991
- moving to model space 212, 216
- plotting last/first 991, 994

PAPERUPDATE system variable 981

paragraphs 799

- selecting 799

paragraphs (multiline text) 804, 811–
812, 852

- format codes 852
- indenting 811–812
 - hanging intents 812
- selecting 804

parallel dimensions. *See* baseline dimensions
parallel lines 645
filleting 645
parallel projections (of 3D views) 173
creating 173
dynamically 173
perspective views
parameter labels 539–540
specifying 539–540
parameter properties 468, 470, 484, 499–501, 503–504, 506, 509, 520, 539–540, 547–548
Base Type property 501, 509
Chain Actions property 484, 539, 547–548
custom properties 468, 470, 539–540
Distance Type property 499–500, 504, 506
Number of Grips property 520
Scale Type property 501, 503
Value Set properties 539
parameter sets 476–477, 518
adding actions to 477
adding to dynamic block
definitions 518
copying 477
customizing 476
properties 476
deleting actions from 477
types 518
parameter values 484, 531, 545–546
associating with other data 531
deleting from value sets 545
extracting 484
incremental values 546
input properties 531
minimum and maximum
values 546
parameters (for dynamic blocks) 468, 470, 477–482, 484, 487–496, 498–500, 502–504, 506–508, 510–516, 518, 520, 523, 531, 539–540
actions supported by 482, 484
adding to dynamic block
definitions 468, 470, 484, 487–496, 518, 520, 523
alignment parameters 492
base parameters 496
flip parameters 493
grips and 520
linear parameters 488
lookup parameters 495
methods 470
overview 468, 484
parameter sets 518
point parameters 487
polar parameters 489
rotation parameters 491
visibility parameters 494, 523
XY parameters 490
assigning actions to 496, 498–500, 502–504, 506–508, 510–516, 520, 531
array actions 513–514
flip actions 511–512
grips and 520
lookup actions 515–516, 531
move actions 498–499
overview 496
polar stretch actions 506–507
rotate actions 508, 510
scale actions 500, 502
stretch actions 503–504
deleting actions from 477
dependencies (dynamic block element relationships) 470, 482
dependency highlighting 479, 481
displaying in Block Editor 478–481
grips. *See* grips (for parameters)
key points 482, 496, 520
labeling 539–540
properties. *See* parameter properties
settings 478
types 482, 484, 498, 520
value sets 478, 482, 484
values. *See* parameter values
Partial Open option (OPEN command) 123

partial plot configuration files. *See PCP files*
partial saves 127, 129–130
partially open drawings 123–125
loading geometry into 125
named objects available in 123
opening 123–124
xref-dependent layers as displayed in 124
Partner Products and Services (Autodesk) 13
parts (discipline-specific) 54–56, 58
accessing 54, 56
downloading 55, 58
parts lists, creating 568
passwords 1053
for DWF files 1053
PASTECLIP command 360
using in dynamic prompt tooltip 360
pasted information 610–611
converting to drawing file format 610–611
PASTESPEC command 610
pasting 610–611, 822
objects, into drawings 610–611
text, from other application 822
PAT file 773
paths 189–191, 695, 697–699, 728–730, 1084
extruding into solids or surfaces 695, 697, 728
linking to cameras 189
lofting to create objects 699, 729–730
motion path animations 189–191
sweeping to create objects 697–698
updating 1084
pattern properties, of tool palette tools 91
PC2 (legacy plotter configuration) files 965–966, 1000
importing settings from 966, 1000
into layouts 966
into plot style tables 1000
PC3 (plotter configuration) files 974, 984
and PMP files 984
tool for creating 974
pc3 editor 1065
PCP (legacy plotter configuration) files 965–966, 1000
importing settings from 966, 1000
into layouts 966
into plot style tables 1000
PDF files 1022, 1044
plotting 1022
plotting sheet sets to 1044
publishing sheet sets to 1044
PEDIT command 666
command options 666
pen plotters 1012–1013
pen assignments 1012–1013
pen settings 966, 1000, 1007, 1012, 1072–1073
color mapping tables for previous release settings 1007
for DWF files 1072–1073
editing 1072–1073
importing PCP/PC2 settings 966, 1000
into layouts 966
into plot style tables 1000
for raster (penless) plotters 1012
pens 1012–1013
assigning to plot styles 1012–1013
settings. *See pen settings*
virtual pens 1012–1013
performance improvement 166, 320, 322–325, 461, 1110–1111
3D display and adaptive degradation
Aerial View window dynamic updating 166
drawing size, reducing 461
fills display 323
hardware acceleration
lineweight display 324–325
lineweights and 320
memory allocation
memory and

object display properties 322
performance tuner log
physical memory
software acceleration
text display 323
tuning manually
xrefs with indexes 1110–1111
performance tuner log
PERIMETER system variable 393
perimeters, calculating 393
period (.) 296, 844
 wild-card character 296, 844
perpendicular lines, drawing 376
perspective views (of 3D views) 171
 defining 171
 turning off
PFACE command 719
physical memory
pickbox cursor 588–589, 597, 599
 resizing 588–589, 599
 selecting objects with 588, 597
PICKFIRST system variable 274
PICKSTYLE system variable 602
pipe character (|) 1106
 dependent named object name
 divider 1106
pixels 607
 removing strays 607
placeholders 823, 826
 inserting fields as 823, 826
plan views 175, 177, 318, 366
 changing 3D views to 177
 using object snaps in 366
 wide polyline linewidth display
 in 318
planar closed spline curves 393
 calculating areas/perimeters 393
planar objects 711
 cutting solids with 711
planar surfaces 705–706
planes 328, 739, 741
 clipping. *See also* clipping planes
 constraining movement to 739, 741
 workplane 328
playing
 animations

PLINE command 359
 dimensional input 359
plines. *See* polylines
plot area 954, 980–981
 setting 954, 980–981
 See also printable area
plot configurations. *See* plotter configurations
plot devices. *See* plotters
Plot dialog box, plot area options 980
plot file drawing sets 1038
 creating and publishing 1038
plot file formats 1019–1020, 1024
 DWF format 1019
 DXB format 1020
 PostScript formats 1024
plot files 1026
 creating 1026
 DWF files. *See* DWF plot files
plot offset (of layouts) 956, 984
 adjusting 956
 paper-saving feature conflicts 984
plot orientation 964, 985
plot scale 957, 985–987
 paper size and 985–986
 setting 957, 985–987
 to fit the paper 957, 986–987
 for layouts 957, 985
 on the Model tab 985
 using a custom scale 987
 using a real-world scale 987
plot scale lists 957, 985
 modifying 957, 985
plot settings 990
Plot Stamp dialog box 991
 displaying 991
plot stamps 975, 991, 1047
 publishing sheets (in sheet sets)
 with 1047
 turning on 991
plot style colors 1009–1010
 assigning 1009–1010
Plot Style control (Properties toolbar) 273
plot style linetypes 1014–1015
 assigning 1014–1015

plot style lineweights 1014
 assigning 1014
Plot Style Manager 975, 999
Plot Style Table Editor 1004
plot style tables 959–960, 962, 975, 988,
 996–1002
 assigning to layouts 998, 1002
color-dependent. *See* color-dependent
 plot style tables
converting color-dependent tables
 into named tables 997
converting drawings to a different
 table type 996–997
creating 999
descriptions, changing 1000
editing for layouts 960
editing plot styles in 1001
folder for 999
importing PCP/PC2/CGF file pen
 settings into 1000
for layouts 959–960
 creating 960
 editing 960
 selecting 959
managing 999
named. *See* named plot style tables
previewing the effects of in
 layouts 999
renaming 1000
rendered viewport plotting options
 and 962, 988
shaded viewport plotting options
 and 962, 988
types 975, 996
 setting 996
 switching 996
plot styles 273, 279, 961, 975, 991, 994–
 997, 1001–1002, 1004, 1008–1017
 assigning to layers 279
 assigning, to objects 273
 in color-dependent plot style
 tables 996
color-dependent. *See* color-dependent
 plot styles
converting drawings to a different
 style type 996–997
converting from color-dependent
 styles to named styles 996–
 997
converting from named styles to
 color-dependent styles 997
dithering 1009, 1011
editing 1001, 1008
fill styles 1016–1017
grayscale conversion 1010–1011
for layers. *See* layer plot styles
in layouts, displaying 961
line end styles 1015–1016
line join styles 1016
linetypes 1014–1015
lineweights 1014
in named plot style tables 1002
named. *See* named plot styles
NORMAL style 1004, 1008
for objects. *See* object plot styles
pen assignments 1012–1013
plotting with/without 991, 994
screening (color intensity) 1009–
 1010
True Color values 1009
types 975, 995
 See also plot style tables
plots 964, 985, 1017–1018
area. *See* plot area
batch. *See* batch plots
orientation 964, 985
previewing 1017–1018
scaling. *See* plot scale
 styles. *See* plot styles
Plotter Configuration Editor,
 starting 951
plotter configuration files 74, 974
 for DWF files. *See* DWF plotter
 configuration files
 specifying the path 74
plotter configuration settings 974
plotter drivers
 See also nonsystem plotter/printer
 drivers
plotter icon (status bar) 1040, 1042
Plotter Manager 974

plotters 948–949, 980–981, 983, 1012–1013, 1045
paper size 981, 983
pen plotters 1012–1013
 pen assignments 1012–1013
publishing sheets (in sheet sets)
 to 1045
raster plotters 1012–1013
 pen settings 1012–1013
reconfiguring 949
selecting 948–949, 980
plotting 188, 205, 210, 217, 318, 323, 325, 758, 790, 957, 974, 976–978, 981, 985, 987, 990–991, 993–994, 1017–1020, 1022, 1024–1028, 1044, 1050, 1064, 1122
background plotting 977–978
 canceling 978
 turning on/off 977
batch plotting
 See also batch plots
camera glyphs 188
checking job status 977
drawings 976
file formats. *See* plot file formats
layouts 217
light glyphs 1122
lineweights 318, 990, 993
 with custom values 318
from the Model tab 205, 981
order of objects in plotting 325
overview 974
with page setup plot settings 978
paper space objects last/first 991, 994
plot files 1026
preparing drawings for 210
 process steps 210
previewing drawings for 1017–1018
published electronic drawing
 sets 1050
raster files 1022
scaling drawings for 205, 957, 985
section objects and 758
sheets in sheet set to PDF files 1044
styles. *See* plot styles

test plot performance 323
text frames 323
to files 1019–1020, 1022, 1024–1028, 1044, 1064
 DWF files 1019, 1027–1028
 DWF6 files 1064
 DXB files 1020
 PDF files 1044
 plot files 1026
 PostScript files 1024–1025
 raster files 1022
 using a custom scale 987
 using a real-world scale 987
viewing job details 978
wipeout objects 790
with/without plot styles 991, 994
plug-ins. *See* standards plug-ins
plus sign (+) 47, 383–384
 tracking points marker 383–384
 tree view level display button 47
plus-or-minus tolerances. *See* lateral tolerances
plus-shaped marker blips 607–608
 removing 607–608
PMP (plot model parameter) files 984
point filters. *See* coordinate filters
point lights 1118–1120, 1123–1125, 1127
 adding 1118–1120
 attenuation rates 1124–1125
 location 1123
 positioning 1127
point markers (for point objects) 389–390
 restyling 389–390
 See also markers
Point Move parameter set 518
point objects (points)
 as reference points. *See* reference points
 See also points (geometric)
point parameters 482, 484, 487, 518, 539, 547–548
actions supported by 482, 484
adding to dynamic block definitions 487

Chain Actions property 484, 539, 547–548
grip type 482
sets 518
pointer input 359, 363
absolute coordinates, specifying with # sign 359
changing settings 363
relative polar coordinates as default for subsequent points 359
pointer input tooltips 361
entering different types of coordinates 361
pointing device buttons
See also pick button (mouse)
pointing devices 75, 242, 430
buttons. *See* pointing device buttons
click and drag actions in tree views 242
restoring the default on startup 75
sketching freehand with 430
points 162, 189, 275, 329, 357, 380–384, 387–388, 392, 670–671, 932, 934
association points (of dimensions) 934
control points 670–671
coordinate locations, displaying 329
coordinate values 329, 392
calculating 392
displaying 329
definition points (of dimensions) 932
displaying coordinate values 275
fit points 670
locating 357
motion path animation and 189
for offset objects 388
specifying 162, 380–382, 387–388
with angle overrides 380
with coordinate filters 381–382
with direct distance entry 387
by locking angles 380
panning by 162
from temporary reference point offsets 388
tracking 383–384
points 383–384
points (geometric) 359, 366, 386, 408, 461, 482, 496, 520, 612–613, 620–621, 780
base points. *See* base points
copying objects using two points 620–621
defining hatch boundaries with multiple points 780
insertion base points 461
key points (on parameters) 482, 496, 520
measuring the distance between two points 408
moving objects using two points 612–613
snapping to points on objects 366
specifying 359, 386
in dimensional input
 tooltips 359
 with tracking 386
tracking to points on objects. *See* object snap tracking
points (point objects) 388–391, 449
creating 449
inserting 388–391
 marking equal segments on objects 388, 390–391
 marking measured intervals on objects 388–389
markers 389–390
sizing 449
styling 449
polar angles 378, 380
specifying 378, 380
tracking along. *See* polar tracking
Polar Array parameter set 518
polar arrays 623–624, 626
 creating 623–624, 626
 in 3D 626
polar coordinates 328, 332, 334, 336–339, 361
 entering 328, 332, 334, 336–339
 absolute 334
 cylindrical 336–337

relative 334
spherical 338–339
entering in dynamic input
 tooltips 361
polar distances 378–379
 drawing objects using 379
 specifying 378–379
Polar Move Pair parameter set 518
Polar Move parameter set 518
polar parameters 482, 484, 489, 518,
 520, 539, 543, 546–548
actions supported by 482, 484
adding to dynamic block
 definitions 489
Chain Actions property 484, 539,
 547–548
grip numbers 520
grip type 482
incremental values 546
minimum and maximum
 values 546
sets 518
value sets 543, 546
polar stretch actions 506–507, 550
adding to dynamic block
 definitions 506–507
parameter supported by 506
redefining in dynamic block
 definitions 550
Polar Stretch Pair parameter set 518
Polar Stretch parameter set 518
polar tracking 376–380, 383
 drawing objects using 379
 and Ortho mode 376–377
 specifying angles 378, 380
 turning on/off 378
PolarSnap 377
 and grid snap 377
 See also polar distances
polling cycle 267
 teamwork on sheet sets 267
polyface meshes 719
 creating 719
POLYGON command 423–424
polygon selection windows 591–592
 crossing polygon selections 591–592
window polygon selections 591–592
polygon windows. *See* polygon selection windows
polygonal clipping boundaries 1094
polygonal meshes. *See* surface meshes
polygons 322, 393, 420, 424–425, 438,
 591–592, 719, 1094
 calculating areas/perimeters 393
 circumscribed polygons 424
 as clipping boundaries 1094
 CPolygons (crossing polygon
 selections) 591–592
 drawing 420, 424–425, 438
 inscribed polygons 424–425
 irregular. *See* closed polylines
 line widths, changing 322
 polyface meshes 719
 WPolygons (window polygon
 selections) 591–592
polyline clipping boundaries 1094
polyline segments 649, 665–666, 668–
 669
 chamfering 649
 joining 665–666, 668
 tapering 669
polylines 189, 316–318, 322, 325, 393,
 419–423, 430, 438–441, 628, 644,
 646, 649, 651, 664–668, 790,
 1094
arc polylines 420, 422, 438, 440
boundary polylines 421, 423, 439,
 441
calculating areas/perimeters 393
chamfering 649, 651
as clipping boundaries 1094
closed 420, 438
converting into lines 423
decurving 667
drawing 419, 421, 430, 438, 440
 freehand 430
 in model space 419
exploding 664
filleting 644, 646
filleting lines with 644
fitting 666
 curves to 666

splines to 666
for wipeout objects 790
joining 665–666, 668
to lines, arcs, or polylines 665–
666, 668
line widths 318, 322, 666
changing 322, 666
setting 318
linetypes 316–317, 667
changing 317
generating 667
setting 316–317
modifying 665, 667
motion path animations and 189
offsetting 628
overlapping, draw order 325
segments. *See* polyline segments
wide. *See* wide polylines
See also objects
polysolids 691–693, 728
creating 692–693
defined 691, 728
Portable Network Graphics files. *See* PNG
(Portable Network Graphics) files
portrait orientation 964, 985
setting 964
positive direction 344
of axes, determining 344
of rotation about axes,
determining 344
PostScript driver 1024
PostScript files 1024–1025
formats 1024
levels 1024
plotting 1024–1025
PostScript fonts, TrueType
equivalents 836
pound sign (#) 295, 359, 361, 814, 823,
844, 868
absolute coordinates entry in pointer
input 359
formula error display 868
invalid field display 823
prefix in pointer input,
accessing 361
stacked text indicator 814
wild-card character 295, 844
precedence of operators and
expressions 404
QuickCalc rules 404
precedence of standards files 144–145
reordering 145
precision 328
drawing tools 328
precision of drawing units 114, 116
setting 114, 116
predefined 3D meshes 717
prefixes (for dimension text) 897
preset views 174
3D views 174
pressing bounded areas of 3D solids 734
preview images (of drawings) 130
saving drawings with 130
previewing 55, 58, 130, 999, 1017–1018
drawings 130, 1017–1018
for plotting 1017–1018
online content 55, 58
plot style table effects in layouts 999
plots 1017–1018
previous release formats (of
AutoCAD) 829
compatibility of fields with 829
previous views 160, 164, 168
restoring 168
zooming to 160, 164
primary units (for dimensions) 898–900
adding and formatting 900
symbol for 898
prime symbol 116
feet specifier 116
primitive solids 728
types of 728
Print button (Help toolbar) 9
printable area 946, 948, 951, 984
modifying 984
paper-saving feature conflicts 984
plotter maximums 951
printers 948–949, 980
reconfiguring 949
selecting 948–949, 980
See also plotters

printing 1058, 1064
3D DWF files 1058, 1064
See also plotting

privacy 17, 55
and Communication Center 17
and DesignCenter Online 55

Procedures tab (Help window) 8
information depth 8

procedures, Help topic searches for 7

product support, Migration Tools 2

productivity. *See* performance improvement

profiles 71, 698–699, 728
extruding to create solids 728
lofting to create solids 699
revolving to create solids
sweeping to create solids 698
workspaces and 71

profiles (of environment settings) 72–73, 75
creating (saving) 72–73
exporting/importing 72
making current 73
before starting the program 73
specifying on startup 75

project names 1083, 1085–1086
adding/removing 1085
defining xref paths with 1083
displaying 1085
modifying 1086

projected tolerance symbol 939

projected tolerances 939
zones 939

projecting 2D images onto 3D object surfaces. *See* image mapping

projection lines. *See* extension lines

projects 1084, 1087
clearing the current project 1087
making current 1084, 1087
See also project names

prompts (in commands). *See* command line prompts

prompts (in commands). *See* dynamic prompts

properties 90–92, 119, 170, 185, 222, 259, 407, 426, 757
camera properties 185
of drawings. *See* drawing properties
of dynamic block elements. *See* parameter properties
of dynamic block elements. *See* action override properties
of layout viewports, modifying 222
modifying with QuickCalc 407
multiline properties 426
of tool palette tools 90–92
changing 90, 92
displaying 91
overrides 91
of views 170
section objects 757
of sheet sets 119
of sheets/sheet sets 259
editing 259
See also layer properties
See also object properties

PROPERTIES command 842

Properties palette 224, 273–275, 402–403, 407, 728, 799, 843–844, 858
accessing QuickCalc from 402–403
changing multiline text 799, 843–844
changing object properties 274–275
displaying object properties 274
expanding/collapsing the properties list 275
manipulating 3D solids and surfaces 728
modifying a layout viewport scale 224
modifying properties with QuickCalc 407
moving multiline text 843–844
opening 274
removing the description pane 274
table properties 858

Properties toolbar 273, 279
property controls 279

property controls (Properties toolbar) 279

Property Lookup Table dialog box 518, 531, 533–534
displaying 518, 531
property types 531, 533
reverse lookup 534
Unmatched row 531

Property Lookup Table dialog box 538
specifying values 538

proxy fonts 833, 837

PS (PostScript) files 1024–1025
format 1024
plotting 1024–1025

PSLTSCALE system variable 232

Publish dialog box 1031, 1038, 1041, 1048
modifying sheet sets for publishing 1048
modifying the sheet list 1031, 1038, 1041
opening from the Sheet Set Manager 1048

publish options 1027–1028, 1047, 1050
saving 1050
setting 1047, 1050
from Sheet Set Manager 1047

Publish Options dialog box 1050
saving the settings 1050

published 3D DWF files 1058, 1064
viewing and printing 1058, 1064

published DWF files 1030, 1050–1051, 1054–1055
canceling jobs in process 1030
checking jobs in process status 1030
including block information
in 1054
including block template information
in 1055
including layer information in 1054
metadata options 1050
output directory for 1051
changing the default 1051
plotting 1050
viewing 1050
viewing job details 1030

published electronic drawing sets 1050
plotting 1050

viewing 1050

published PDF files 1050
metadata options 1050

publishing 261–262, 758, 971, 977–978, 1027–1029, 1031, 1037–1038, 1040, 1042–1043, 1045–1047, 1049

3D DWF Publishing and 1029
in the background. *See* background publishing

background publishing 978
canceling 978

checking job status 977

drawing sets 1031, 1037–1038, 1040, 1042, 1049
electronic sets 1040, 1042
paper sets 1037–1038
plot file sets 1038
republishing 1049

electronic drawing sets 1027–1028
as DWF files 1028
for distribution 1027

section objects and 758

sheet sets 261–262, 1043, 1045–1047
to DWF files 1043, 1045
including plot stamps 1047
option settings 1047
to plotters 1045
using page setup overrides 1046

sheets (in sheet sets) 971
and applying named page setup
as override 971

viewing job details 978

pulling bounded areas of 3D solids 734

purging 155–156, 282, 290, 311, 585, 607, 609

block definitions 585

layers 155–156, 282, 290
unreferenced layers 282, 290

linetypes 311

named objects 607, 609
linetypes 609

pyramids 681, 689–690, 728

3D solids 681
as primitive solids 728

creating 689–690
pyramid frustum 689–690

Q

QDIM command 879
QSELECT command 594
quadrant grips 655
queries (in databases)
 stored. *See* stored queries
queries (in Help). *See* natural language
 queries (in Help)
question mark (?) 296, 844
 wild-card character 296, 844
Question Mark button (in dialog
 boxes) 10
Quick Help (Info palette) 11
 displaying 11
 locking 11
 navigating 11
 printing information 11
Quick Select dialog box, filtering
 dimensions 878
Quick Select, filtering selection sets 594
Quick Setup wizard 110–111
 starting drawings 111
Quick Text mode, turning on/off 323
QuickCalc calculator 397, 399–404, 406–
 409
 absolute value function 400
 accessing 401–403
 from within a command 402–
 403
 directly 401, 403
 from the Properties Palette 402–
 403
 basic math functions 400
 changing size and appearance 399
 clearing the History area 406
 clearing the Input box 399
 displaying coordinate values 408
 displaying the angle of a line 408
 displaying the intersection of two
 lines 409
 as an expression builder 397
 functions 397

layout 397
measuring the distance between two
 points 408
memory functions. *See* QuickCalc
 memory
modifying properties in the Properties
 palette 407
rounding numbers 400
scientific functions 400
truncating numbers 400
units display and handling
 rules 404
variables. *See* calculator variables
See also History area
See also Input box
QuickCalc memory 401
 adding the current value to the stored
 value 401
 clearing the stored value 401
 restoring the stored value 401
 storing a value in 401
quotation marks 6
 Help topic search-text delimiters 6

R

radial dimensions 876, 915, 918–919
 creating 915, 918–919
 creating centerlines automatically
 with 919
 types 876, 915
radians 410
 converting degrees to 410
 converting to degrees 410
radius dimensions 876, 915, 918, 932
 creating 915, 918
 definition points 932
 foreshortened. *See* jogged radius
 dimensions
 See also dimensions
rapid decay area (of spotlights) 1124
raster driver 1021
raster files 1021–1022
 creating (plotting) 1022
 size 1021

raster images 80, 85, 130, 1068
boundaries. *See* raster image boundaries
creating tools for 80
paths. *See* raster image paths
preview images 130
resolution 1068
using a tool created from 85
raster plotters 1012–1013
pen settings 1012–1013
RASTERPREVIEW system variable 130
rays 450–451
about 450–451
drawing 451
raytraced renderer. *See* Photo Raytrace (renderer)
Readme topic (in Help) 10, 20
Real face style
real-time panning. *See* dynamic panning
real-time zooming. *See* dynamic zooming
real-world scales 987
plotting using 987
Realistic visual style 178
rearranging 95, 101–102, 216
layout tabs 216
tool palette groups 101
tool palette tools 95
tool palettes 102
in tool palette groups 102
reassociating 250, 254, 934–935
dimensions 934–935
moved sheets to sheet sets 250, 254
recent input history display 29
recording 732–733
animation
history for composite solids 732–733
records. *See* database records
RECOVER command 134
recovering 137, 244
sheet set data (DST) files, previous versions of 244
from system failures 137
See also restoring
recovery audits 134
recovery files 137
name format 137
RECTANG command 359, 423–424
relative Cartesian coordinates for subsequent points in pointer input 359
rectangles 322, 424–425
drawing 424–425
line widths, changing 322
rectangular areas. *See* rectangular windows
rectangular arrays 623
creating 623
rectangular meshes 717, 720
rectangular selection areas, selecting objects in 590
rectangular window clipping
boundaries 1094
rectangular windows 160, 163, 590, 1094
as clipping boundaries 1094
as selection areas 590
zooming to 160, 163
red error outline, in dynamic input
 tooltips 361
REDO command 605
redoing actions 605–607
reducing 278
visual complexity 278
REFCLOSE command 1101
Refedit toolbar 1098–1099
reference 640–641
scaling objects by 640–641
Reference Manager, about 1084
reference points 388, 449
drawing 449
offsetting points from 388
See also point objects
references 1096
reflection (of light)
diffuse. *See* diffuse reflection
specular. *See* specular reflection (highlights)
reflection color (of materials)
See also specular color
reflection lines 511

refreshing the image of a block tool on a tool palette 92

regardless of feature size value (S/RFS) 938

regenerating 320

 drawings 320

 lineweight display and 320

regions 393, 452–454

 adding 453–454

 calculating areas/perimeters 393

 combining 452, 454

 composite regions 452, 454

 creating 452–454

 out of boundaries 452, 454

 intersections 453–454

 subtracting 453–454

 uses 452

regular polygons 424–425

 drawing 424–425

regular polygons. *See* polygons

relative coordinates 328, 331–332, 334–338, 361, 612, 614, 621

copying objects using 621

entering 328, 331–332, 334–338

 Cartesian 331–332, 335–336

 polar 332, 334, 337–338

entering, in pointer input

 tooltips 361

moving objects using 612, 614

relative paths, for xrefs 1082

relative polar coordinates 359

 as default for subsequent points in pointer input 359

Reload option (XREF command) 1090

reloading 1090

 xrefs 1090

relocated xrefs, notification of 1083

relocating UCSs 340, 344

 in 3D space 344

REMEMBERFOLDERS system

 variable 122

removing 83, 145, 248–249, 254, 576, 579, 584–585, 588–589, 591–592, 603, 607–609, 732–733, 772, 777, 788, 992, 1032, 1034, 1039, 1041, 1085, 1099–1100

blips 607–608

block attributes 576, 579

block definitions 584–585

drawing sheets (from drawing sets) 1032, 1034, 1039, 1041

hatch associativity 772, 788

hidden lines 992

 plotting from layout tabs 992

 plotting from the Model tab 992

hidden lines. *See* removing hidden lines in 3D objects

history from 3D solids 732–733

islands (from hatch areas) 777

objects 588–589, 591–592, 603

 from groups 603

 from selection sets 588–589, 591–592

objects, from working sets 1099–1100

project names 1085

sheet subsets 248

sheet view categories 248

sheets (from sheet sets) 249, 254

standards files from drawings 145

stray pixels 607

tool flyouts 83

unused named objects 607, 609

 linetypes 609

See also deleting

renaming 100, 128, 169, 187, 215, 249, 263–265, 291–292, 301, 343, 528, 757, 970, 1000, 1006–1007, 1032, 1035, 1039, 1042, 1107

cameras 187

drawing sheets (in drawing sets) 1032, 1035, 1039, 1042

drawings named with

 double-byte/special characters 128

layer settings 301

layers 291–292

layouts 215
named objects 1107
named page setups 970
named plot styles 1006–1007
plot style tables 1000
section objects 757
sheet selections 263–265
sheets (from sheet sets) 249
tool palette groups 100
UCSs 343
views 169
visibility states 528
rendered images 988–989
 rendering before plotting 989
 viewport plotting options 988–989
 aliasing in. *See* anti-aliasing
rendering 758
 section objects and 758
 See also Render (renderer)
renderings. *See* rendered images
renumbering 249
 sheets (from sheet sets) 249
reordering objects in groups 603–604
repairing 134–135
 damaged drawing files 134–135
 open drawings 135
repeating commands 27, 34, 37
 in the command window 37
 by right-clicking 27
replacing 844–845, 928, 930
 text 844–845, 928, 930
 dimension text 928, 930
 multiline text 844–845
REPORTERROR system variable 138
republishing 1049
 drawing sets 1049
resaving all sheets in sheet sets 268
RESETBLOCK command 562
 resetting more than one block
 reference at a time 562
resetting 35–36, 562
 block references 562
 system variables 35–36
 transparently 35–36
 view settings
reshaping objects 638, 734
resizing 25, 37, 40–41, 45, 64–65, 166–167, 220, 362, 588–589, 599, 638, 846, 857
Aerial View window image 166–167
the command window 37, 40–41
command window 40–41
DesignCenter 45
layout viewports 220
leaders 846
objects 638
pickbox cursor 588–589, 599
tables 857
toolbars 25, 65
 tooltips 362
windows 64
resolution 191, 990, 1022, 1068–1069
 animation 191
 of DWF files 1068–1069
PDF files 1022
for plotting shaded images 990
 See also precision
Resource Drawings tab (Sheet Set Manager) 241
restarting recently used command 35
restoring 112–113, 136–137, 168–169, 181, 196, 198, 263, 284, 299, 301, 340–342, 350, 352, 607–608, 788, 925
the default drawing template
 file 112–113
default visual styles 181
dimension styles 925
drawings 136–137
 from backup files 136–137
erased objects 607–608
hatch associativity 788
layer settings 284, 299, 301
model space viewports 196
named views 168–169
previous views 168
saved views 169
sheet selections 263
UCSs 340–342, 350, 352
 named UCSs 342, 350
 previous UCSs 340–341, 352
 to the WCS 341

viewport arrangements 198
See also recovering
See also repairing
return button (mouse)
 See also right mouse button
REVCLOUD command 455
reverse lookup 534, 537
reverse quote 296, 844
 wild-card character 296, 844
revision clouds (markups) 455–457
calligraphy style 455
converting closed objects to 456
converting objects to 455
creating 455–456
 from scratch 456
creating with calligraphy pen
 style 456
editing lengths of arcs or chords 457
normal style 455
setting default values for arc
 lengths 455, 457
styles 455
REVOLVE command 704
revolved meshes 716, 721
revolved solids 729–730
revolved surfaces 729–730
REVSURF command 716, 721
right mouse button
 See also right-clicking
right-clicking 26–28, 34
 customizing 26, 28
 displaying shortcut menus 27
 in the drawing area 27
 executing commands 34
 repeating commands 34
 repeating commands by 27
 time-sensitive behavior 26, 28
 turning on 28
right-hand rule 344
roll-over behavior 32, 45, 88–89
 dashboard 32
 DesignCenter 45
 Tool Palettes window 88–89
rolling windows up or down 64
rotate actions 508–510, 514
 adding to dynamic block
 definitions 508, 510, 514
Base Type property 509
parameter supported by 508
rotate grip tool 735, 741
rotated dimensions 909, 913
 extension line origins 909
 See also dimensions
rotating 175, 235, 339, 345, 373, 375,
 615–618, 656, 658, 736, 741–742,
 928–929
 3D solid subobjects
 3D views 175
cameras
dimension text 928–929
grid 375
grip tools and 736
objects
 in 3D 616
 to an absolute angle 616–617
 around an axis 618
 constaining rotation 742
 by dragging 615
 grip tools and 741
 with grips 656, 658
 by a specified angle 615
 See also mirroring
snap angle 373, 375
UCSs 339, 345
views 235
 in layout viewports 235
rotation angle 85, 341
 prompting for, when placing block or
 xref from tool palette 85
 UCSs 341
rotation direction about axes, positive,
 determining 344
rotation grips 520, 561
rotation parameters 482, 484, 491, 518,
 539, 544, 546–548
 actions supported by 482, 484
 adding to dynamic block
 definitions 491
 Chain Actions property 484, 539,
 547–548

- grip type 482
incremental values 546
minimum and maximum
 values 546
sets 518
value sets 544, 546
- Rotation Set parameter set 518
rotation snaps, copying objects
 with 660, 662
- rounding numbers (in QuickCalc) 400
rounding off dimension values 901–902
rounding off drawing units 116
rounding. *See* filleting
rounds. *See* fillets
rows (in array actions) 513
 offset 513
- rows (in database tables). *See* database
 records
- rows (in tables) 857–863, 869–870
 adding 859
 averaging values in 869
 counting cells in 870
 deleting 860
 formatting 861–863
 header rows 862
 modifying 857–859
 setting number of 857
 summing the values in 869
 title rows 862
- RTF files 822–823
 importing 822
 inserting 822–823
- rubber-band line 376, 386
 in tracking 386
 in orthogonal locking 376
- ruled meshes 714, 720
- ruler (Multiline Text Editor) 811–812
- RULESURF command 714, 720
- running object snaps 365–366, 368–370
 cycling through 368
 overriding 369–370
 setting 366
 turning on/off 365–366
- S**
- S (regardless of feature size symbol) 938
saved views 169
 restoring 169
- saving 71–73, 95, 102–103, 127–130,
 136, 144, 167–168, 191, 198,
 236, 238, 262, 268, 281, 299–
 300, 342, 350, 431, 551–552,
 596, 759, 865, 871, 1050, 1110–
 1112
- animation files 191
backup files 129, 136
 when saving drawings 129, 136
- block definitions 551–552
- block definitions as drawing
 files 130
- drawings
- automatic saves 129, 136
 to different formats 127
 incremental saves 127
 with layer indexes 1110–1112
 partial saves 127, 129–130
 with preview images 130
 reducing the time required
 for 127
- saving backup files when 129,
 136
- See also* exporting drawings
 with spatial indexes 1110–1112
- environment variables. *See* profiles (of
 environment settings)
- filtered lists 596
- freehand lines 431
- layer filters 281
- layer settings 299–300
- layout templates 236, 238
- named views 167–168
 settings saved 167
- profiles 72–73
- publish options 1050
- section objects 759
- selected objects 127, 129
- sheet selections 262
- sheets in sheet sets, resaving all 268
- standards files 144

table changes 865, 871
tool palette groups 103
tool palettes 95, 102
UCSs 342, 350
viewport arrangements 198
views 167–168
 settings saved 167
workspaces 71
saving back 1101–1102
 in-place reference editing
 changes 1101–1102
scale actions 500–503
 adding to dynamic block
 definitions 500, 502
 base point type 501, 503
 parameters supported by 500
 Scale Type property 501, 503
SCALE command 114–115
 converting drawing units 114–115
scale factors 205, 223, 639–640, 888
arrowheads 888
dimensions. *See* dimension scale
for layout viewport views 223
for plotting 205
scaling objects by 639–640
scale locking 218, 224
 in layout viewports 218, 224
Scale Type property 501, 503
 for scale actions 501, 503
SCALELISTEDIT command 223, 957, 985
SCALETEXT command 847
scaling 81, 114, 204–207, 223–224, 231–232, 315, 541, 558, 639–641, 656, 659, 842, 846–847, 856, 859, 957–958, 985–987, 1014–1015
3D solid subobjects
annotations 205
blocks 81, 205, 207, 558
dimensions
 See also dimension scale
drawings 114, 204–205, 957, 985–987
 to fit the paper 957, 986–987
 for plotting 205, 957, 985
dynamic blocks 541
fill patterns 1015
hatch patterns 205
inserted content 207
leaders 846
linetypes 205, 231–232, 315, 1014–1015
 in layout viewports (paper space) 231–232
 non-ISO linetypes 1015
lineweights 958
multiple text objects 847
objects 639–641, 656, 659
 by reference 640–641
 by scale factor 639–640
 with grips 656, 659
plots. *See* plot scale
table columns/rows 859
tables 856
text 205–206, 842, 847
views 223–224
 in layout viewports 223–224
scientific functions (in QuickCalc) 400
 using 400
screen display 62, 324, 607
 cleaning up 607
 clearing 62
 resolution. *See* resolution
 updating 324
screening (color intensity) 229–230, 1009–1010
 of objects in layout viewports 229–230
ScriptPro (Migration Tool) 2
scripts 69, 74, 87
 creating command tools for
 executing 87
 running, on startup 74
 turning on animation while script runs 69
SDF (space-delimited format) files 571
Search dialog box (DesignCenter) 51
 loading the content area from 51
search paths 74, 103, 122–123, 131–132, 1082–1083, 1086
 adding/deleting 1086

specifying 132
 for support files 132
specifying (changing) 74, 103, 122–
 123, 131, 1082–1083, 1086
 for the default drawing
 folder 122–123
defining with project
 names 1083
for plotter configuration
 files 74
for support files 74, 131
for tool palette files 103
 type options 1082
for support files 131–132
Search tab (Help window) 3, 5–7
 advanced search rules 6
 check boxes 7
Search tool (Windows) 130, 844
 finding drawing files 130, 844
Search-the-Web link (natural language
 queries) 7
searching 3, 5–7
 Help topics (for words/phrases) 3,
 5–7
 advanced search rules 6
 basic search rules 5
 in the current topic 3
 See also finding
second grips 755
Section Boundary state 754
SECTION command 750
section objects 750–754, 756–759, 761–
 762
 about 751
 cameras and 758
 creating 751–752
 defined 750
 grips 754
 jogged segments 753, 759
 live sectioning 761
 multiple section objects 762
 object states 754, 757
 orthographic sectioning 753
 plotting 758
 properties 757
 publishing 758
 rendering 758
 saving 759
 shortcut menus 756
 viewing 758
 views and 758
Section Plane state 754
Section Volume state 754
sectioning 750–754, 756–759, 761–763
 2D sections 763
 3D sections 763
 3D solids 750
 cameras and 758
 creating sections
 cross sectioning 750
 cutaway geometry 762
 grip actions 754
 jogged segments 753, 759
 live sectioning 750–751, 761
 object states 754, 757
 publishing or plotting section
 objects 758
 section objects 750–752
 shortcut menus 756
 views and 758
SECTIONPLANE command 750
sectored views 123
security of files 1053
 password security for DWF
 files 1053
segment end grips 755
SELECT command 597
Select Template dialog box 108
 displaying 108
 starting drawings in 108
selected objects 127, 129
 saving 127, 129
selecting 37, 279, 287–288, 565, 602,
 655, 734, 799, 804, 824, 856,
 858, 863, 865
3D solid subobjects
attributes 565
fields (in text) 824
grips 655
groups 602
individual solids in composites 734
layers 279, 288

multiple layers 287
objects. *See selecting objects*
paragraphs 804
table cells/columns 856, 858
tables 863
text 37, 799, 804, 865
 in tables 865
words 804
selecting objects 588–594, 596–599, 655
 by classification property 594
 close together 588
 customizing selection 596
 cycling through objects 588–589
 after entering an editing
 command 597
 before entering an editing
 command 597
 with grips on 655
 individually 588–589
 in irregular areas 591–592
 methods 591–592
 methods (default options) 598–599
 changing 599
 multiple objects 590
 noun/verb selection 589, 597–598
 with the pickbox cursor 588, 597
 preventing selection 593
 in rectangular areas 590
 with selection fences 591–592
 with selection windows 590
selection areas 590–591
 irregular 591
 rectangular 590
selection fences 590–592
 selecting objects with 591–592
selection highlighting 597–598
 previewing effects. *See selection*
 previewing
 turning off 598
selection methods 591–592
 listing options 591–592
selection previewing 588, 597–599
 cycling through objects 588
 turning on/off 588, 599
selection sets 518, 588–589, 591–592,
 594–595, 656–657, 738
 3D solid subobjects in
 for actions. *See action selection sets*
 adding 588
 objects to 588
 adding objects to 595
 assigning actions to 518
 creating, with Quick Select 594
 excluding objects from 595
 filtering 594
 in 3D views 738
 limiting grip display in 656–657
 named. *See groups*
 removing 588–589, 591–592
 objects from 588–589, 591–592
selection windows 590
 creating 590
 crossing selections 590
 window selections 590
 See also polygon selection windows
selections 590–592
 crossing 590–592
 polygon selections 591–592
 window polygon selections 591–592
 window selections 590
 sending objects to back 325
 draw order of objects 325
 separating 3D solids into original
 shapes 744–745
servers
 See also directory servers
setenv AutoLISP function, using to set the
 MaxHatch system registry
 variable 772, 787, 789
settings. *See system variables*
setup wizards 110–111
 starting drawings with 111
shaded images 178
 customizing shading
 customizing shading
 visual styles and 178
 See also shading
shaded UCS icon 356

shaded viewports 962–963, 988–990
plotting options 962–963, 988–990
 Rendered option 989
resolution 990
setting 989

SHADEMODE command 1098

shading *See* shaded images

Shadow Display property

shadows 1123
 hardware acceleration and
 lighting and 1123
 visual styles

sheet creation templates 243, 262
 creating sheet sets from 262
 specifying 243

sheet list (Publish dialog box) 1031, 1038, 1041
 modifying 1031, 1038, 1041

sheet list (Sheet Set Manager) 247–248
 creating subsets in 247
 removing subsets from 248
 reordering sheets in 247

Sheet List tab (Sheet Set Manager) 241

sheet list tables (on title sheets) 251, 258
 creating 258

sheet properties, published in DWF files 265

sheet selections (named sheet sets) 75, 262–265
 deleting 263–265
 loading on startup 75
 modifying 264
 renaming 263–265
 restoring 263
 saving 262

Sheet Set control (Sheet Set Manager) 241

sheet set data (DST) files 242, 244
 backing up/recovering 244

sheet set data backup (DS\$) files 244

Sheet Set Manager 119, 240, 244, 247, 266, 1043, 1047–1048
 interface controls 240
 lock icons for sheet sets 266
 opening 244

opening the Publish dialog box
 from 1048

organizing sheet sets 119

reordering sheets in 247

setting publish options from 1047

Sheet Set Manager window 75
 not displaying on startup 75

sheet sets 119, 240, 242–246, 249–250, 253–254, 259–262, 266–268, 971, 1043–1048, 1063
 archiving 261
 closing 245
 creating 242–244, 249, 253, 262
 from drawing files 244
 from example sets 243
 new sheets 249, 253
 preparation steps 243
 from sheet creation
 templates 262
 custom properties 259–260
 creating 260
 eliminating multiple layout tabs
 for 243
 lock icons in Sheet Set Manager 266–267
 false 267
 modifying for publishing 1048
 opening 245
 opening drawings within 249, 253–254
 organizing 119, 246
 page setups for 243, 262
 overriding 243, 262
 plotting 1044
 properties 119, 259
 editing 259
 publishing 261–262, 1043, 1045–1047, 1063
 to DWF files 1043, 1045
 including plot stamps 1047
 option settings 1047
 to plotters 1045
 to 3D DWF files 1063
 using page setup overrides 1046
 reassociating sheets that were moved
 to another folder 250, 254

removing sheets from 249, 254
renaming sheets in 249
renumbering sheets in 249
resaving all sheets in 268
status data for team members using
 sheets 266
team use of 266
transmitting over the Internet 261
updating sheets for team members not
 using Sheet Set
 Manager 267
 using named page setups 971
sheet subsets (in the sheet list) 246–248,
 259, 1045
 creating 247
 properties 259
 editing 259
 publishing, to DWF files 1045
 removing 248
sheet view categories 246, 248, 259
 creating 248
 properties 259
 editing 259
 removing 248
sheet views 249–252, 255–258
 adding 250–251, 255
 callout blocks for 250, 256–258
 components (entities) 251
 drawing files for 249
 label blocks for 250, 257
 removing 252
sheets (in sheet sets) 240, 243, 247, 249,
 251–254, 258–260, 265–268, 971
 accessed by multiple users through
 different logical drives 266
 applying named page setups as
 override when publishing
 sheets 971
 common operations on 249
 creating 249, 253
 custom properties 259–260
 creating 260
 importing layouts as 252
 modifying 249
 multiple layouts from single drawing
 files as 243

properties 259, 265
 editing 259
 in published DWF files 265
 removing from sheet sets 249, 254
 renaming in sheet sets 249
 renumbering in sheet sets 249
 reordering in the sheet list 247
 resaving all sheets in sheet sets 268
 specifying same output options in
 named page setups 971
 status data for team members 266
 title sheets 251, 258
 creating sheet list tables on 258
 updating for team members not using
 Sheet Set Manager 267

shelling 745
 solids 745

SHIFT+A 369
 overriding running object snaps 369

SHIFT+click 588–589
 adding objects to selection sets 588
 removing objects from selection
 sets 588–589

SHIFT+right click 365
 displaying the Object Snap shortcut
 menu 365

SHIFT+right-click 367
 displaying the Object Snap shortcut
 menu 367

SHIFT+TAB keys 865
 moving to the previous cell 865

shifting the XY plane 351

shortcut functions 412
 predefined calculator variables 412

shortcut menus 26–28, 242
 customizing 26–27
 displaying 26–28, 242
 customizing right-click
 behavior 26, 28

options typical of 26

turning off 27
 in the drawing area 27

Show button (Help toolbar) 10

SHOWHIST system variable 732

shrinking views. *See* zooming

SHX (font) files 798, 805, 833, 836–837
bullet character 805
custom fonts 837
proxy fonts for 833, 837
Unicode character support 798
silhouette edges
single-line text 794–797, 832, 840–843
 aligning 795, 797
 compressing 795
 creating 795–796
 editing 842
 fields in 795
 objects, modifying 842–843
 obliquing angle 840
 orientation 841
 styles 795, 797, 832
 not affecting 832
 specifying 797
 See also text
single-sheet DWF files 1051
 specifying 1051
sizing 449
 point objects 449
SKETCH command 429–430
sketching (freehand lines) 429
sky (3D modeling) 63
 display options 63
sky horizon 63
 display options 63
sky zenith 63
 display options 63
slash (/), stacked text indicator 814
slicing solids 710–711
slide-out panel 32
smooth faces
smooth lighting
smoothing fonts 802
snap (snap grid)
 See also PolarSnap
snap (snap grid) 371, 373–375, 377
 and PolarSnap 377
 overriding 373, 375
 overview 371
 rotating angle 375
 snap grid spacing 373–374
 turning on/off 374
snap angle 373
 rotating 373
snap base point 373
 changing 373
Snap mode. *See snap*
snapping to points on objects 366
 See also snap (snap grid)
snaps
 See also object snaps
software acceleration
solid fills 323–325, 780, 788
 creating 780
 modifying 788
 overlapping objects, draw order 325
 turning off 323–324
 See also 2D solids
 See also donuts
 See also Fill mode
 See also solid-filled hatches
 See also wide polylines
solid hatch patterns
 See also gradient fills
solid models 680
 creating 680
 See also 3D solids
solid primitives 681
 creating
 types of 681
solid-filled hatches 781, 788
 creating 781
 modifying 788
SOLIDEDIT command 746
 cleaning and checking solids 746
 editing solids with
solids 729–730, 744–745
 creating 729–730
 with REVOLVE 729–730
 separating into original shapes 744–745
solids, 3D. *See* 3D solids
sorting layers 295, 298
 using wild-card characters 295
source applications
 See also server applications
space-delimited format (SDF) files 571

SPACEBAR key 34
executing commands 34
repeating commands 34
SPACETRANS command 847
spatial indexes 1110–1112
saving drawings with 1110–1112
special characters 128, 800–801, 832
inserting in multiline text 800–801
for naming objects 832
renaming drawings named with 128
See also Special Characters section in this index
specular color (of materials)
See also specular reflection (highlights)
specular reflection (highlights)
See also specular color
speed of animation 190
spelling. *See* checking spelling
spheres 681, 688–689, 728
3D solids 681
as primitive solids 728
creating 688–689
spherical coordinates, entering 338–339
spirals 448, 672
drawing 448
modifying 672
spline curves
See also spline-fit polylines
spline leaders, creating 819
spline-fit polylines 446, 635, 666
converting into splines 446
creating 446, 666
trimming and extending 635
splined polylines. *See* spline-fit polylines
SPLINEDIT command 670
command options 670
splines (NURB curves) 628
offsetting 628
splines (NURBS curves) 189, 393, 446,
670–672
calculating areas/lengths 393
control points 670–671
converting 446
spline-fit polylines into 446
drawing 446
editing 670, 672
with grips 670
fit points 670
motion path animations and 189
order of 671
refining the shape 671
tolerance 446, 670
spotlights 1118–1120, 1123–1125, 1127
adding 1118–1120
attenuation rates 1124–1125
beam angle 1124
falloff angle 1124
field angle 1124
hotspot angle, setting 1124
location 1123
positioning 1127
rapid decay area 1124
square brackets 296, 844, 898–899
alternate units symbol 898–899
wild-card character 296
wild-card characters 844
square brackets with hyphen 844
wild-card characters 844
square brackets with tilde 844
wild-card characters 844
square feet units (in QuickCalc) 404
square-bracketed hyphen, wild-card
character 296
square-bracketed tilde, wild-card
character 296
squares 424–425
drawing 424–425
squeezing text 795
stacked text (multiline text) 814–817,
852
creating 814–815, 852
format codes 852
editing 816
unstacking 817
Stand-Alone Installation Guide,
accessing 2
standard grips 520, 561
STANDARD style 426
multilines 426

standards (for named objects) 142–144, 146–147, 153–154
converting layers to 153–154
maintaining in collaborative environments 142, 146–147
named object types for which standards can be created 142
setting 143–144
 standard options 143
violations. *See* standards violations
See also standards files

standards audits 142, 146–147, 149, 152
batch audits 147, 152

standards check files 147, 151
 creating 151
 opening 151
 overrides for, specifying 151
See also standards files

standards files 142, 144–145, 147
 associating with drawings 144
 conflicts between 144
 creating 144
 file name extension 144
 precedence 144–145
 reordering 145
 removing from drawings 145
 saving 144
 uses for 142
See also standards check files

standards plug-ins 143–144, 146
 specifying 143–144, 146

standards violations 142–143, 146–150, 152
 checking drawings for 142, 146, 149, 152
 fixing 143, 147–149
 ignored problems display 147, 150
 turning on/off 150
 notification alerts 146, 148, 150
 turning on/off 148, 150
 types 142

starting 4, 73–74, 76, 107–112, 150
 Batch Standards Checker 150
 Developer Help 4

drawings 74, 107–112
 with the default drawing template file 109–110
 with drawing templates/template files 108–112
 with templates 74
 with wizards 111

Help 4

program 73–74, 76
 making environment profiles current before 73
 making profiles current before 73
 using command line switches 74, 76

static coordinate display 329

status bar 28–29, 328, 1040, 1042
 button display 29
cursor coordinate display 29

cursor location coordinate display 328

icon and notifications display 28

plotter icon 1040, 1042

STB file name extension 975

stray pixels 607
 removing 607

stretch actions 503–504, 506, 550
 adding to dynamic block definitions 503–504
 Distance Type property 504, 506
 parameters supported by 503
 polar stretch actions 506
 redefining in dynamic block definitions 550

STRETCH command 639

stretch frames 503, 506, 550
 polar stretch actions 506
 redefining 550
 specifying 503

stretching 613, 615, 639–640, 656–658, 795, 846
 leaders 846
 moving objects by 613, 615
 objects 639–640
 objects, with grips 656–658
 text 795

See also extending styles
visual. *See* visual styles

subdividing model space viewports 195

subobjects (3D solids). *See* 3D solid edges

subobjects (3D solids). *See* 3D solid faces

subobjects (3D solids). *See* 3D solid vertices

Subscription Center 13–14

- accessing 14
- installing 14
- services available 13

substitute fonts 838

SUBTRACT command 453, 708–709

subtracting 394, 397, 453–454

- areas 394, 397
- regions 453–454

suffixes 897

Summary tab (Drawing Properties dialog box) 120

summing the values in table cell ranges 869

sunlight, simulating 1117–1118, 1127–1129

- about 1117–1118, 1127
- brightness 1128
- sun angle 1129
- sun color 1129
- turning sun on/off 1127

support files 131–132

- search paths. *See* search paths
- specifying search paths for 131–132

support folders, specifying on startup 74

surfaces 680, 696–699, 701, 704–707, 711, 713, 728–730

- creating 680, 697–699, 701, 704–707, 728–730
- 3D solids from 707
- by lofting 699, 701, 729–730
- by revolving 704, 729–730
- by sweeping 697–698, 728
- for 3D models 680
- from objects 704–706

extruding objects into 696, 728

faceted. *See* meshes

faceted. *See* surface meshes

lofting 729–730

manipulating 728

mapping 2D images onto. *See* image mapping

- meshes 713
- normals 730
- slicing solids with 711
- sweeping 728

See also faces

surveyor angles 118

- specifying 118

SWEET command 448, 695, 697–698

See also EXTRUDE command

sweeping 728

- creating objects 728

switching 70–71

- workspaces 70–71

swiveling the camera

symbol libraries 52, 54

See also block libraries

- accessing 54
- loading the DesignCenter content area with 52

symbol library drawings. *See* block library drawings

symmetrical tolerances 903, 905

syntax of expressions 404

- evaluated in QuickCalc 404

system failures 135, 137

- recovering from 137
- repairing damaged drawing files due to 135

system printers, paper size 950

system variables 10, 35–36, 39

- affecting dialog boxes 39
- entering on the command line 36
- help on 10
- listing 36
- resetting 35–36
- transparently 35–36

T

tab characters, in attribute extraction template files 573

TAB key 368, 865
cycling through running object snaps 368
moving to the next cell 865
Table Cell Format dialog box 864
table of contents. *See* sheet list tables (on title sheets)
Table Style dialog box 861, 864
table styles 860–861, 863
applying 863
defined 860
defining new styles 861
reapplying 863
TABLEEXPORT command 860
TABLEINDICATOR system variable 872
tables 567, 825, 856–865, 867–872
adding/deleting columns or rows 859–860
averaging the values in cell ranges 869
background color 862
block attributes in 567
cell and row indicator display 872
background color control 872
turning off/on 872
cell margins 862
cells. *See* cells (in tables)
columns. *See* columns (in tables)
counting the cells in cell ranges 870
creating 857
defined 856
direction of 861
editing cells 856
editing text in 864, 867
entering formulas in 868, 871
entering text in 864–865
exporting 860
formatting text in 861–865
gridlines 863
header rows 862
height/width, changing 856
inserting blocks in 864, 867
inserting fields in 825, 868
merging cells 858–859
number of columns/rows in 857
Properties palette and 858
resizing with grips 857
rows. *See* rows (in tables)
saving changes 865, 871
selecting text in 865
summing the values in cell ranges 869
table styles 860–861, 863
text color 862
title rows 861–863
Tablet mode 430
sketching in 430
tabs (multiline text), setting 811
TABSURF command 715, 721
tabulated meshes 715, 721
tapered angles 696
extruding solids from 696
tapering 420, 439, 669, 782
wide polyline segments 420, 439, 669
wide polylines 782
targets 188–189
camera targets defined
changing for cameras 188
linking to paths 189
taskbar (Windows) 126
displaying drawings on 126
TASKBAR command 126
teams 266–267
limited member access to DST files
when not using Sheet Set Manager 267
logical drives for sheet set work 266
member access to DST and DWT files 266
sheet set use 266
status data for sheets in current sheet set 266
work sessions polled for status changes 267
work with sheet sets with Sheet Set Manager 266
work with sheet sets without Sheet Set Manager 267
template drawings. *See* drawing templates/template files

templates 243, 262, 288
 creating 288
 sheet creation templates 243, 262
temporary files 131, 136
 name format 136
 specifying location 131
temporary override keys 371
 changing keyboard response time
 for 371
test plots, performance 323
testing 471, 550
 dynamic blocks 471, 550
text 37, 205–206, 323, 325, 629, 794–
 795, 799, 802, 804, 822, 830,
 840–842, 844–845, 847, 852, 861–
 865
 adding hyperlink fields to 830
 compressing 795
 draw order 325
 changing 325
 entering 864–865
 in table cells 864–865
 expanding (stretching) 795
 finding 844–845
 multiline text 844–845
 formatting 861–865
 in table cells 861–865
 importing 822
 mirroring 629, 842
 obliquing angle 840, 852
 format code (multiline
 text) 852
 orientation 841
 overlapping, draw order 325
 pasting, from other
 application/AutoCAD
 files 822
 replacing 844–845
 multiline text 844–845
 scaling 205–206, 847
 selecting 37, 799, 804, 865
 multiline text 804
 in tables 865
 simplified display 323
 smoothing fonts 802
 types 794

wide (multiline text), format
 code 852
text alignment 795, 802–803, 852
 in single-line text 795
 in multiline text 802–803, 852
 vertical alignment format
 codes 852
 See also text justification
text boxes, for leader text 821
text editors (multiline text) 850–851
 alternate 850–851
 creating text 851
 editing text 851
 formatting text 851
 specifying 850
 default 850
text fields. *See* fields (in text)
text files 822–823
 importing 822
 inserting 822–823
text frames, displaying/plotting 323
text height 804, 838–839, 847, 852
 converting between model space and
 paper space 847
multiline text 804, 852
 changing 804
 format codes 852
 setting 838–839
text justification, multiline text 802–
 803, 848
text objects 841–843, 847–848
 changing justification without
 changing location 848
 grips 842
 modifying 841–843
 single-line text objects 842–843
 moving 842
 scaling 842, 847
text obliquing angle 840, 852
 setting 840, 852
text orientation, setting 841
text styles 795, 797–798, 803, 824, 831–
 834, 838–841, 862, 897–898
 assigning fonts to 833–834
 creating 832
 deleting 832

for dimension text 897–898
in fields 824
font styling 833
height in 838–839
modifying 832
for multiline text 798, 803
 overriding 803
naming 832
obliquing angle in 840
orientation in 841
for single-line text 795, 797
 specifying 797
standards. *See* standards (for named objects)
styles not affecting
 single-line/multiline text 832
 in tables 862
text width 795, 852
 compressing/expanding 795
wide text (multiline text), format code 852
text window 37–38, 64
 closing 38
 copying all text in 37
 copying text to the command line 38
 displaying 38
 font selection 64
 selecting (highlighting) text in 37
switching from the command window to 37
TEXTFILL system variable 833
TEXTTOFRONT command 325
thawing layers 225–228, 284
 all layers 284
 in layout viewports 225–228
 in paper space 228
THICKEN command 707
thickness 724–725
 adding to objects 724–725
tick marks 482, 542, 562
 displayed at location of valid block reference values 562
 displaying in value sets 482, 542
tilde (~) 296, 844, 849
 incorrect word flag 849
 wild-card character 296, 844
tiled viewports. *See* model viewports
time-sensitive right-clicking
 behavior 26, 28
 turning on 28
tints
 faces
title rows in tables 861–864
title sheets 251, 258
 creating sheet list tables on 258
tolerance 446, 670
 of splines 446, 670
tolerance symbol (\pm) 903
 entering 903
tool flyouts 80, 83–84, 93
 adding/removing 83
 customizing 84
 restoring default settings 93
 using tools on 83
tool palette files 102–103
 default path 103
 file name extension 102
 lock icon 103
tool palette groups 31–32, 99–103
 adding tool palettes to 99
 copying tool palettes between 101
 creating 99, 101
 within other groups 101
 dashboard and 31–32
 deleting 100
 displaying 99
 rearranging 101
 rearranging tool palettes in 102
 removing tool palettes from 99
 renaming 100
 saving 103
tool palette tabs, moving 95
tool palette tools 80–82, 84–87, 89–95
 command tools 86
 creating command tools 86–87
 creating from objects 80–81, 94
 icon display style 89–90
 icon updates 91
 images, changing 93

modifying 95
properties 90–92
 changing 90, 92
 displaying 91
 overrides 91
rearranging 95
refreshing the image of block
 tools 92
using command tools 87
using tools created from
 blocks/xrefs/raster
 images 85
using tools created from
 dimensions 82
using tools created from gradient
 fills 84
using tools created from hatches 84
using tools created from objects 82
tool palettes 50–51, 79–81, 86, 94–96,
 98–99, 101–103, 467, 473, 557,
 770
 adding drawings from 51
 adding items to 50
 adding to tool palette groups 99
 adding tools to 80–81, 94
 command tools 86
 copying between tool palette
 groups 101
 creating 50–51, 94, 96
 with DesignCenter content 50–
 51
 deleting 95
 displaying all 102
 dragging hatch patterns from 770
 exporting 102
 importing 102
 inserting blocks with 467, 557
 lock icon 95
 opening blocks from 473
 organizing 98
 organizing blocks with 467
 rearranging 102
 in tool palette groups 102
 removing from tool palette
 groups 99
 saving 95, 102

tabs, moving 95
tools. *See* tool palette tools
use in product versions other than the
 one created in 103
See also Tool Palettes window
Tool Palettes window 79, 81, 88–90
 displaying 81
 settings 88–90
 icon display style 89–90
 rollover behavior 88–89
 transparency 88–89
Tool Properties dialog box 770
 hatch pattern options 770
toolbars 24–25, 45, 65, 67, 69, 471,
 1098–1099
 Block Editor toolbar 471
 closing 25
 DesignCenter toolbar 45
 displaying 24, 65
 docking 24
 in workspaces 69
 locking/unlocking 65, 67
 Refedit toolbar 1098–1099
 resizing 25, 65
 undocking 25, 65
 for workspaces
tools 328
 precision drawing tools 328
tools, on tool palettes. *See* tool palette tools
TOOLTIPMERGE system variable 62
tooltips 62, 358–359, 361–364, 368
 appearance 62
 AutoSnap 368
 coloring 362
 coordinate display in 359
 dimensional input 359
 displaying prompts in 364
 in Dynamic Input 358
 dynamic prompts, choosing options
 for 362
 entering coordinate values in 361
 merging drafting 363
 resizing 362
 specifying points in 359
 transparency control 362
 turning on/off 62

See also dimension input tooltips
See also dynamic prompts (tooltips)
See also pointer input tooltips

TOOLTIPS system variable 62

tori 681, 691, 728

- 3D solids 681
- as primitive solids 728
- creating 691

tracking 386

- to offset points, specifying points with 386

tracking points 383–384

- on objects 383–384

transmitting sheet sets (over the Internet) 261

transparency 40, 64, 88–89, 362, 757, 798

- command window 40
- faces
- In-Place Text Editor 798
- multiline text objects 798
- section plane indicator 757
- Tool Palettes window 88–89
- tooltips 362
- windows 64

transparent command character (apostrophe) 35

transparent command prompt indicator 35

transparent commands 35

- commands that can be used as 35

tree view 44, 46–47, 242, 287, 294

- DesignCenter 44, 46–47
 - about 44
 - displaying 46
 - level display buttons 47
- Layer Properties Manager 287, 294
 - hiding 287
- Sheet Set Manager 242
 - click and drag actions 242

TRIM command 632, 635

trimming 632, 634–637, 643, 646, 649, 651, 779, 788

- chamfered objects 649
- filleted objects 643
- hatches 779, 788

objects 632, 634–637, 646, 651

- in 3D 635, 637
- chamfering without 651
- while extending 634
- filleting without 646

spline-fit polylines 635

wide polylines 634

True Color colors 303–304, 306–307

- changing 306–307
 - layer colors 306
 - object colors 307
- setting (for new objects) 304

True Color values, for plot styles 1009

TrueType fonts 798, 805, 833, 836–837, 839

- appearance 833
- assigning to text styles 833
- bullet character 805
- PostScript font equivalents 836
 - as SHX proxy fonts 833
 - SHX proxy fonts 837
 - text height values 839
- Unicode character support 798

truncating numbers (in QuickCalc) 400

turned off layers 278

two-color gradient fills 785

- creating 785

U

U command 605

UCS command 471

- in Block Editor 471

UCS icon 62, 356–358, 471

- 2D UCS icon 356
- 3D UCS icon 356–357
- Block Editor drawing area icon 471
- broken pencil icon 357
- changing 357–358
 - appearance of 357–358
- coloring 62
- cross in 356
- displaying, at the origin 356, 358
- shaded UCS icon 356
- styles 356–357
- switching between 357

turning on/off 358
UCS origin 340, 350, 356, 358
defining in 2D 340
defining in 3D 350
displaying the UCS icon at 356, 358
UCSICON command 356–357
UCSs (user coordinate systems) 235,
 339–346, 349–352, 354, 356
2D operations 339
3D operations 343
assigning to viewports 354, 356
defining 339–340, 345–346, 350–
 351
 in 3D space 346
 with specific axes 350–351
 with three points 345
deleting 343
drawing planes for 343, 349
 specifying 343
elevation in 349
moving 339
in multiple viewports 346
origin. *See* UCS origin
paper space limitations on 350
preset systems 349, 352
 selecting 352
relocating 340, 344
 in 3D space 344
renaming 343
restoring 340–342, 350, 352
 named UCSs 342, 350
 previous UCSs 340–341, 352
 to the WCS 341
rotating 235, 339, 345
 around axes 345
 views with 235
rotation angles 341
saving 342, 350
uses 346
UCSVP system variable 346, 354–355
unbounded hatches 779
 creating 779
underground (3D modeling) 63
underlining multiline text 804, 851
 format codes 851

UNDO command 605
 command line options 605
undocking 25, 39–40, 45, 65
 command window 39–40
 DesignCenter 45
 toolbars 25, 65
undoing 1101
 in-place reference editing
 changes 1101
undoing actions 35, 291, 605–607
 canceling commands 35, 606
 changes to layer settings 291
 erasings 605
 most recent action 606
 redoing undoings 605–607
 single actions 605
 specific numbers of actions 606
UNION command 453, 708
units of measurement 328, 404, 409
 converting 409
 coordinate unit types 328
 QuickCalc display and handling
 rules 404
unloading 311, 1109–1110
 linetypes 311
 xrefs 1109–1110
unlocking layers 593
Unmatched row (Property Lookup Table
 dialog box) 531
unreferenced layers 155–156, 282, 290
 purging 155–156, 282, 290
unstacking text 817
UP ARROW key, displaying recent input
 in dynamic prompts 360
updating 50, 53, 166–167, 567–568,
 574–577, 579–580, 754, 829, 879,
 1078, 1090–1091
of the Aerial View window
 image 166–167
associative dimensions 879
attribute extraction table 567–568
block attributes 576–577
block definitions 50, 53, 574–575
block references 576, 579–580
fields 829
multiple fields 829

section object states 754
of viewports, in the Aerial View window 167
xrefs 1078, 1090–1091
updating, the display 324
upgrading the program 2

V

validating 746–747
 3D solids 746–747
Value Set properties 539
value sets (for parameters) 478, 482, 484, 542–546
 deleting values from 545
 in Block Editor 478, 484
 specifying 542–544, 546
 tick mark display 482, 542
values (in QuickCalc) 400, 405–408
 changing the font color in the History area 406
 copying from the History area 406–407
 copying from the Input box 405
 displaying absolute values 400
 displaying coordinate values 408
memory functions. *See* QuickCalc memory
 pasting to the command line 406
 rounding numbers 400
 truncating numbers 400
values of parameters. *See* parameter values
variable attributes 563
 inserting blocks with 563
Vault client software 15
 overview 15
Vault Data Management Server 15
 overview 15
Vault Office Add-in 16
 overview 16
Vault. *See* Autodesk Vault
vector graphics 1068
 resolution 1068
 in DWF6 files 1068

vertical bar (!) (pipe character) 1106
 dependent named object name divider 1106
vertical dimensions 876, 909–910
 creating 910
 extension line origins 909
vertical text 841
 setting 841
vertices 666, 674–675
 of multilines 674–675
 of polylines 666
video cards
video format 191
view box (Aerial View window) 164
view list (Sheet Set Manager) 248
 creating categories in 248
 removing categories from 248
View List tab (Sheet Set Manager) 241
View Plot Area option (Page Setup dialog box) 954, 980
view scale (in layout viewports) 223
 changing 223
view scale lists 223
 modifying 223
View Support Requests option, Subscription Center 13
view transitions 62, 69
 appearance 62, 69
 changing 69
view windows 160, 163
 zooming to 160, 163
viewing 758, 1028, 1030, 1050, 1058, 1064
 3D DWF files 1058, 1064
 drawings
 in 3D
 DWF files 1028, 1050
 in Autodesk DWF Viewer 1050
 published electronic drawing sets 1050
 published job details 1030
 section objects 758
viewing commands 224
 scale locking and 224
viewpoint coordinate 176
 defining 3D views with 176

viewpoints (in 3D space) 175
defining 3D views from 175
viewport arrangements
(configurations) 171, 195, 197–198, 222
default options 195
deleting 198
listing 171, 198
naming 198
placing into layouts 222
restoring 198
saving 197–198
 settings saved 197
viewports 167, 179, 194, 285, 354, 356, 1118
 assigning UCSs to 354, 356
 layers and 285
 lighting in 1118
 updating of, in the Aerial View window 167
 visual styles and 179
views 62, 69, 74, 123, 159–160, 167–170, 173, 176–177, 218, 223–224, 233, 235, 751, 758, 764
3D. *See* 3D view
aligning, in layout viewports 233
boundaries of 169
changing 159, 167
cross sections 751
defining with VPOINT 176–177
 AEC convention 176
 mechanical design
 convention 177
displaying a view on startup 74
flattened views of 3D objects 764
modifying 218, 223
 in layout viewports 218, 223
naming 168
panning 160
in paper space 173
properties 170
restoring 168–169
 named views 168–169
 previous 168
 previous views 168
 saved views 169
rotating 235
 in layout viewports 235
saving 167–168
 settings saved 167
scaling 223–224
 in layout viewports 223–224
section objects and 758
sectored views 123
transitions settings 62, 69
 changing 69
virtual pens 1012–1013
 specifying pen numbers 1013
visibility 218, 225, 278–279, 1079
 of dynamic block geometry. *See*
 visibility parameters
 of dynamic block geometry. *See*
 invisible objects (in dynamic
 blocks)
 of dynamic block geometry. *See*
 visibility states (for dynamic
 block geometry)
of layers 218, 278–279
 in layouts 218, 279
of layers, in xrefs 1079
of objects 225, 278
 in layout viewports 225
layer control 278
visibility parameters 478, 482, 484, 494, 518, 523
actions supported by 482, 484
adding to dynamic block
 definitions 494, 523
grip type 482
sets 518
 specifying 478
Visibility Set parameter set 518
visibility states (for dynamic block
 geometry) 469, 523, 525–530
 changing list order 530
 creating 523, 526–527
 default state 530
 deleting 527
 drawing geometry and 469
 enabling 525
 making current 527
 making objects invisible in 526, 529

making objects visible in 528–529
renaming 528
showing/hiding geometry for 523,
530
See also visibility parameters
VISRETAIN system variable 1079
visual complexity, reducing 278
visual styles 178, 180–181, 736, 989
 3D interactive views
 adaptive degradation and
 backgrounds
 customizing
 default styles 178
 defined 178
 edge display
 exporting 180
 face styles
 faces
 grip tool display and 736
 importing 180
 plotting 989
 restoring default 181
 shading
 shadows
 storing on tool palettes 180
Visual Styles Manager 178
 about 178
VPOINT command 176–177
 defining views with 177
 mechanical design
 convention 177
 defining views with (AEC
 convention) 176
VPORTS command 197
VTOPTIONS command 62, 69
 controlling appearance of transitions
 between views 69
 and transitions between views 62

W

walking through layers 280
walking through models
 settings
 simulating
 starting walk mode

WBLOCK command 127, 129–130, 465,
574
 saving block definitions 127, 130
 saving selected objects 127, 129
WCS (world coordinate system) 339,
341, 345, 430
axes 339
entering coordinates relative to 345
mapping coordinate systems on paper
 to 430
restoring the UCS to 341
web 7
 natural language queries 7
 See also Internet
websites 10
 Autodesk website 10
wedges 681, 683, 728
 3D solids 681
 as primitive solids 728
 creating 683
wide polylines 318, 323, 420, 422, 439,
441, 634, 669, 782, 788
 creating 782
 drawing 420, 422, 439, 441
 lineweight display 318
 modifying 788
 simplified display 323
 tapering 782
 tapering segments 420, 439, 669
 trimming and extending 634
wide text (multiline text), format
 code 852
width 857, 859
 of columns in tables 857, 859
wild-card characters 280–281, 295–296,
844
 filtering layers by name using 280–
 281, 295
 finding text using 844
 naming objects using 296
 sorting layers by name using 295
Window Plot Area option (Page Setup
 dialog box) 954–955, 981
window polygon selections 591–592
window selections 590

windows 62, 64–67, 88, 590, 857
anchored 64, 67, 88
application window 62, 66
docked 88
docking 64
font selection 64, 66
inserting tables in 857
locking 65, 67
resizing 64
rolling up/down 64
selection windows 590
transparency control 64
Windows Clipboard. *See* Clipboard (Windows)
Windows Explorer 119, 122, 822–823
and drawing properties 119
inserting text/RTF files from 822–823
opening drawings in 122
Windows metafiles. *See* WMF files
Windows taskbar 126
displaying drawings on 126
wipeout objects 790–791
creating 790
frames, turning on/off 791
plotting 790
wireframes 636–637, 722–723, 989
creating 723
extending objects in 636
extracting edges and 723
overview 722
plotting 989
tips for designing 722
trimming objects in 637
ZY plane and 723
wizards 110–111
setup wizards 110
starting drawings with 111
WMF files 610–611
converting to drawing file
format 610–611
format 610
WMFBKGND system variable 610
WMFFOREGND system variable 610
words 799, 804, 848–850
adding/deleting to/from
dictionaries 848–850
selecting 799, 804
working sets (of reference objects) 1097, 1099–1100
adding objects to 1099–1100
fading of objects not in 1097
removing objects from 1099–1100
workplane 328
workspaces 69–71, 75
changing settings 70–71
creating 71
customizing
loading with command line
switch 75
profiles and 71
saving 71
selecting a sample 70
switching 70–71
world coordinate system (WCS)
See also WCS (world coordinate system)
World Wide Web. *See* Internet
WPolygons (window polygon selections) 591–592

X

X axis 62, 339, 351, 371
color of 371
coloring 62
in UCSSs 351
WCS 339
XATTACH command 1079
XCLIPFRAME system variable 1080, 1093
XFADECTL system variable 1097
XLOADCTL system variable 1109, 1112
XLOADPATH system variable 1112
XML reports
See also batch audit reports
XREF command 1079, 1090
command line options 1090

xref copies (for demand loading) 1111–1113
creating 1111–1112
setting paths for 1112–1113
Xref icon 1079
exclamation point 1079
xref log files 1109
turning on/off 1109
Xref Manager, displaying 1079
xref paths 1082–1084, 1086, 1089, 1104, 1112–1113
absolute paths 1082
changing 1104
no path option 1083
no path search order 1083
relative paths 1082
specifying (changing) 1082–1084, 1086
defining with project names 1083
type options 1082
for temporary xref copies 1112–1113
xref tools 80
prompt for rotation angle 80
xref-dependent layers 124
as displayed in partially open drawings 124
xrefs. *See* external references (xrefs)
XTP file name extension 102
XY Array Box Set parameter set 518
XY Move Box Set parameter set 518
XY Move parameter set 518
XY parameter 503
Scale Type property 503
XY parameters 482, 484, 490, 500, 506, 518, 539, 544, 546–548
actions supported by 482, 484
adding to dynamic block definitions 490
Chain Actions property 484, 539, 547–548
Distance Type property 500, 506
grip type 482
incremental values 546

minimum and maximum values 546
sets 518
value sets 544, 546
XY plane (construction plane) 351, 723
shifting 351
views of. *See* plan views
wireframe modeling and 723
XY plane (workplane) 328
XY Stretch Box Set parameter set 518

Y

Y axis 62, 339, 351, 371
coloring 62, 371
in UCSSs 351
WCS 339
yellow alert icons 550–551

Z

Z axis 62, 339, 350
coloring 62
in UCSSs 350
in the WCS 339
Z values (3D coordinates), default values 335
zero suppression 902–903, 905–906
in dimension values for lateral tolerances 902–903, 905–906
ZOOM command 160
command line options 160
zoom factor 320
and linewidth display 320
Zoom Original command 168
Zoom Previous command 168
zooming 69, 160–164, 166–167, 373
with Aerial View 164, 166–167
by dragging 162
to the drawing extents 161, 163
to the drawing limits 161, 163
dynamic (Realtime option) 161
grid display and 373
to previous views 160, 164
real time interactive mode

to rectangular areas (view
windows) 160, 163

turning on animation for 69

